3 THE VOLTAGE DIVIDER

The assembly of complex power system models for study using electromagnetic transients simulation tools can be a challenging and trying task, even for the experienced power system engineer. It is highly recommended that the user begin by assembling a very simple circuit.

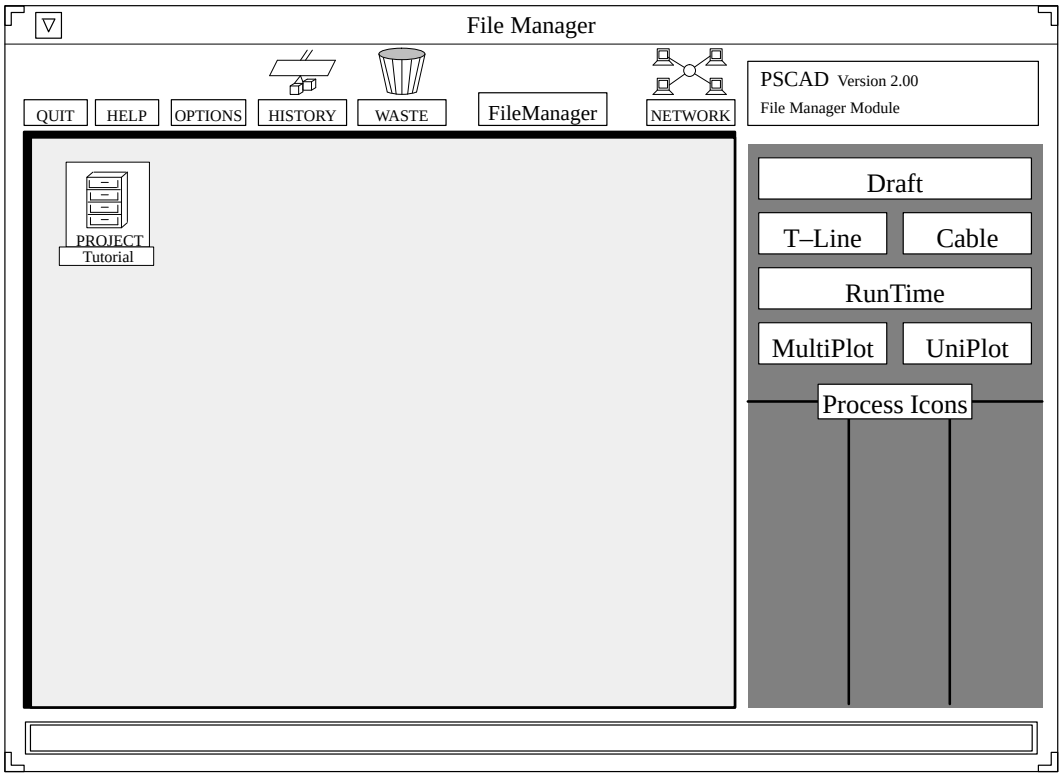
In this section, the user will be guided through all of the steps required to model the simple voltage divider circuit.

3.1 STARTING THE PSCAD FILEMANAGER

To start the PSCAD FILEMANAGER, simply enter the following command in any Unix shell. Note that the host workstation must be running an X11 based Window Manager in order for the PSCAD software to successfully start.

```
% PSCAD &↵
```

A window such as that illustrated below will appear on the screen.



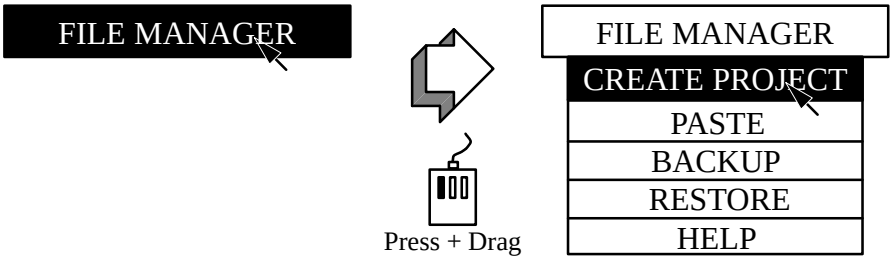
The “Tutorial” icon, in the top left corner of the large open area, is a project which contains completed cases from this *Tutorial Manual*. Users may wish to refer to these in case of difficulty, but are encouraged to try to complete these example cases on their own.

3.1.1 Creating a Project

The FILEMANAGER module allows the user to organise various simulation data into Projects. A project will generally contain all the information on one particular study.

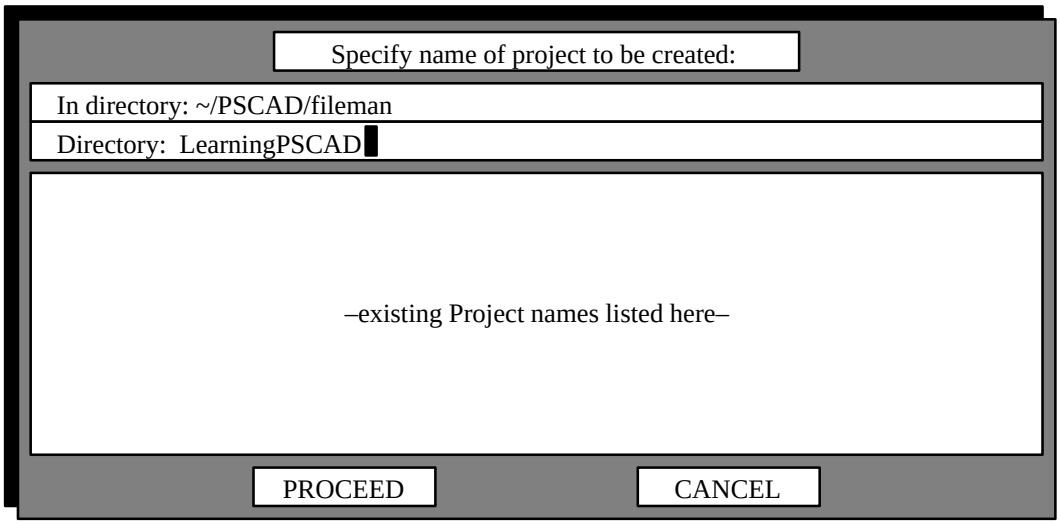
In this manual, the user is "studying" the PSCAD software itself, so a Project with an appropriate name, such as "`LearningPSCAD`", should be created.

To create a new "Project" directory, select the CREATE PROJECT option from the FILE MANAGER menu.



Filenames are "case–sensitive". That is, 'ABC', 'Abc', and 'abc' are all valid and unique names.

This will open a "File Requestor" form, where the name of the Project to be created may be entered. The names currently used by other projects (eg, "`Tutorial`") are also shown in the form, to prevent accidental reuse of an existing Project name.

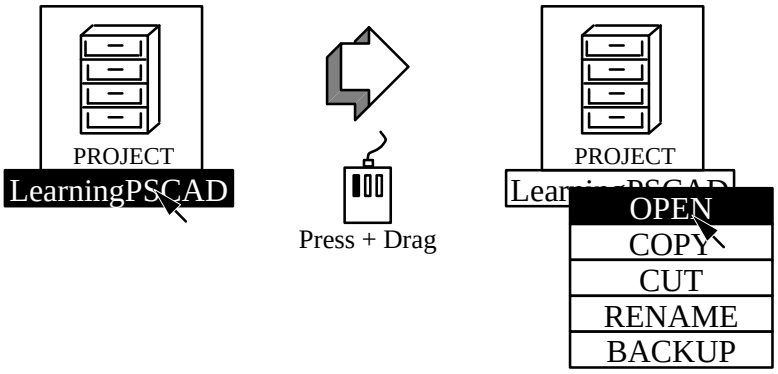


Once PROCEED is selected, the form disappears and a new Project icon appears on the FILEMANAGER window. The FILEMANAGER places the icon such that it will not obscure other existing icons.

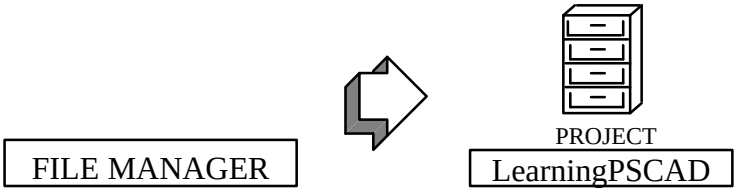
A newly created icon has a dotted border until the user has manually positioned the icon by dragging it to the desired position.

3.1.2 Opening a Project

The user may enter the newly created Project by selecting the OPEN option from the pop–up menu associated with the Project icon.



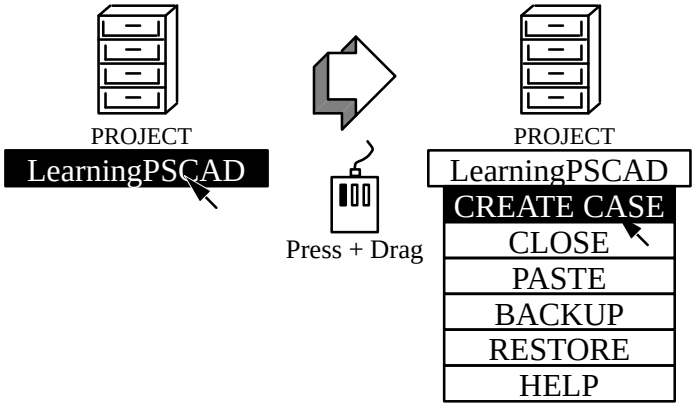
The FILE MANAGER menu button at the top of the FileManager changes to indicate that a project has been opened.



3.1.3 Creating & Opening a Case

The first circuit that will be used to demonstrate the usage of the PSCAD modules is a simple voltage divider circuit. Thus, inside the “LearningPSCAD” Project, a “VoltageDivider” case should be created.

Select the CREATE CASE option from the Project menu.



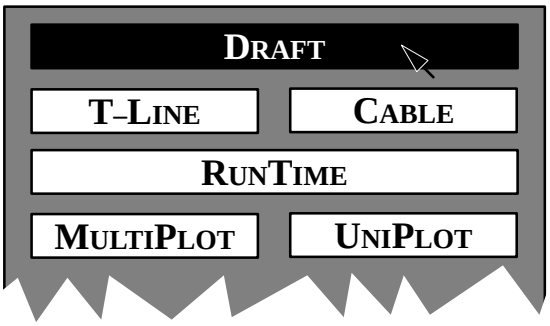
A File Requestor will appear, where the name of the Case to be created is entered. When PROCEED is selected, the form disappears and a new Case icon appears on the FileManager window.

The user can access the newly created Case by selecting the OPEN option from the pop–up menu associated with the Case icon. The menu button at the top of the FileManager will change to reflect the fact that a Case has been opened.

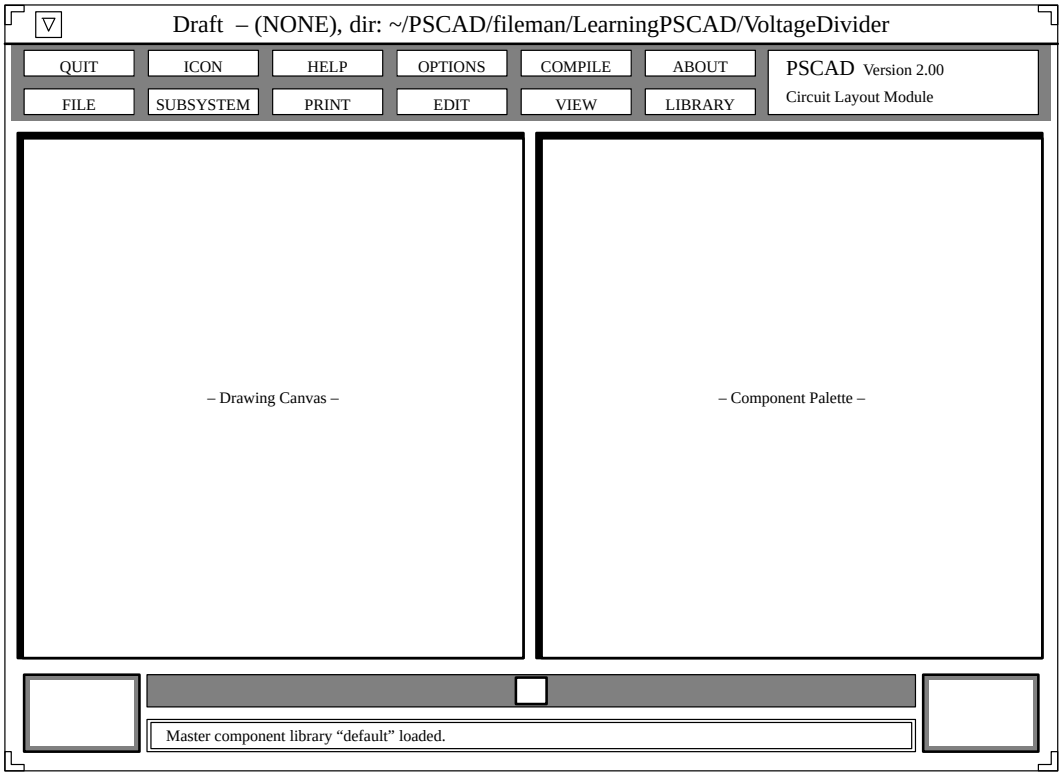
3.2 CREATING THE CIRCUIT DIAGRAM

The PSCAD/DRAFT module is used to "sketch" a pictorial representation of a power system that is to be simulated. From this pictorial representation, the DRAFT module creates datafiles which will be used in future steps of the simulation.

To start the DRAFT module, select the DRAFT button from the right hand side of the FILEMANAGER window.



It may take a few seconds before a new window, similar in appearance to the one shown below, is displayed on the workstation's monitor.



If any problems are encountered in this section of the *Tutorial Manual*, the reader is advised to consult the appropriate sections in the *DRAFT Manual*.

3.2.1 General Sketching Techniques

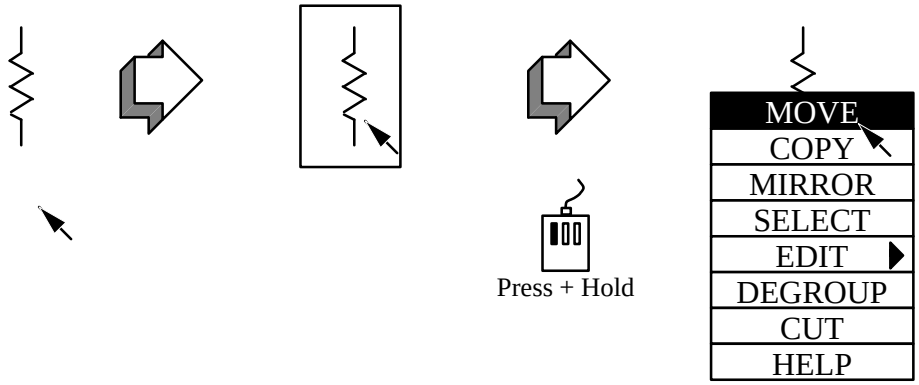
3.2.1.1 The Drawing Canvas and Component Palette

Press + Drag

A sketch of the user's power system model may be made by selecting component icons from the right hand side of the DRAFT window (referred to as the palette) and placing them on the left side of the DRAFT window (referred to as the canvas). Both the palette and canvas can be scrolled by moving the cursor into the desired window area and holding down the right mouse button while moving the mouse.

3.2.1.2 The Component Menu

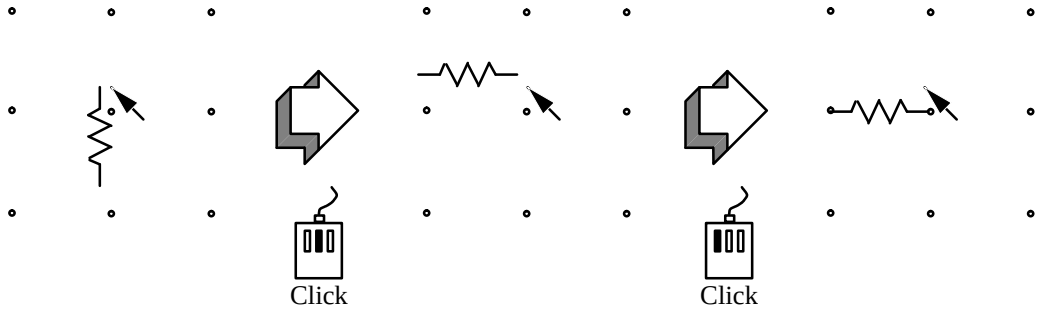
When the cursor is placed on top of a component icon, a solid lined box will appear around that component, signifying that it is currently selected. Depressing the left mouse button while a component is selected will display a menu on the screen as shown below.



3.2.1.3 Copying and Moving Components

Copying a component from the component palette to the drawing canvas will automatically open the edit window for that component. This feature may be disabled from the Options form.

The MOVE and the COPY menu items will cause the highlighted component, or a duplicate of the highlighted component respectively, to "stick" to the cursor until the user clicks the left mouse button. While the user is dragging a component, it can be rotated 90° by clicking the middle mouse button. Subsequent clicks of the middle mouse button again rotate the component icon by 90°. When the component is dropped in the canvas area it will automatically be aligned to the nearest grid dot.

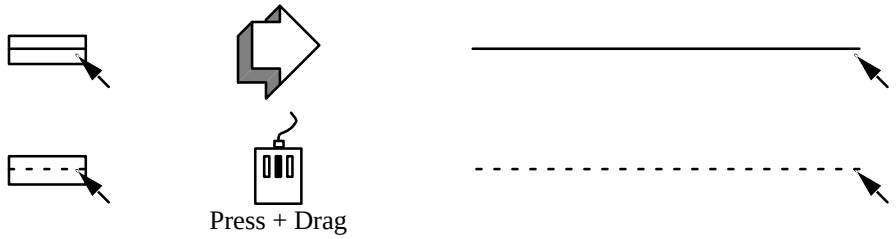


It is possible to scroll the palette and canvas windows using the right–most mouse button while still dragging a component.

3.2.1.4 Stretching Wires and Lines

This will not work if the component is attached to the cursor. "Drop" the component first.

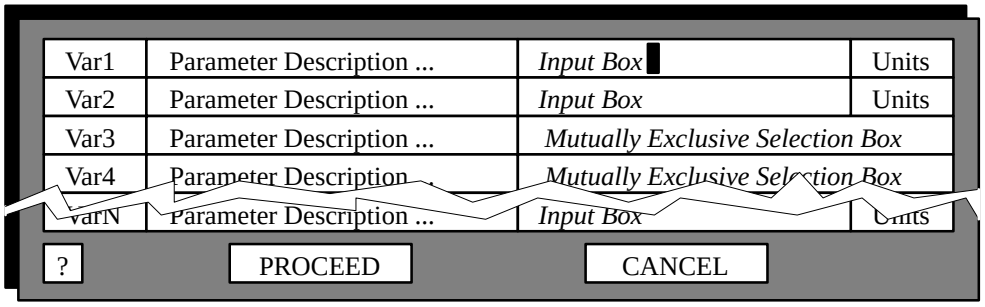
There are two components which are scalable in length. The bus work component icon (a solid line), and the dotted line both may be stretched by holding down the middle mouse button while they are selected and dragging the mouse. Release the mouse button when the component has reached its desired size.



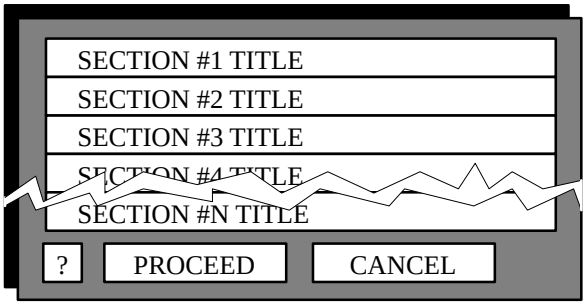
The bus work icon is used to interconnect other component icons. The dotted line is used only for visually highlighting areas of a circuit, and it does not affect the power system model created by the user on the canvas.

3.2.1.5 Editing a Component

When the EDIT menu item is selected, or a component is copied from the palette to the canvas with "Auto Edit" turned on, the "Edit Form" for that component will appear.



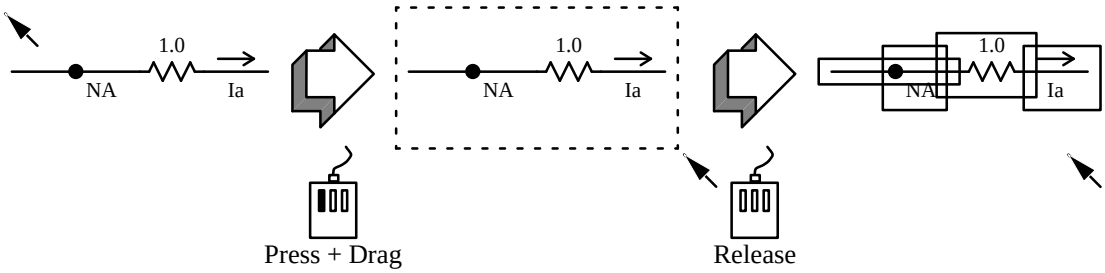
If there are many parameters associated with a component, or if the parameters are better suited to being displayed in groups, a "Section Form" will appear instead. Clicking on a section box will open the respective Edit Form.



With most components, a ? button will be present, from which help on that particular component may be obtained.

3.2.1.6 Working with Group Components

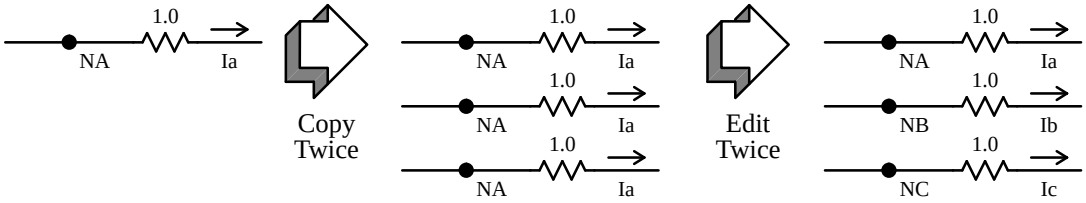
Several components may be collected together to form a "Group Component." To form a group component, "select" two or more components using the SELECT menu item on the component menus, or drag a selecting box around the desired components.



The components which have been selected are surrounded by boxes, to indicate their selection. The selected components are formed into a group component by selecting the GROUP menu item from the top EDIT menu.

Group components can be copied, moved, and rotated like any other component. All components within a group maintain their relative position and orientation when a group is moved or rotated.

Group components are very useful in constructing 3 phase circuits. Phase A of the circuit would be constructed as normal. These components would then be grouped together, and copied as one component to form phase B and phase C. The Phase B and Phase C group components may then be edited to correct any phase dependent information.



When editing a component group, each sub–component's Edit Form or Section Form will appear in turn, and a *Stop* button will be displayed in the lower right corner of each form. Selecting the *Stop* button will *Cancel* the changes to the current sub–component, and stop the editing cycle. Changes which were made to previous sub–components in the editing cycle will not be undone.

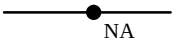
To break a group component into its respective sub–components, select the DEGROUP item from the component menu. If the group component contains any groups as sub–components, the sub–component groups will not be degrouped.

3.2.1.7 Electrical and Dataflow Wires

There are two different classes of "wires" in DRAFT. A wire may be an electrical wire or a dataflow wire. Rather than being an editable property of a wire, the wire class is determined by what the wire is connected to. Wires of different classes should not be inter-connected.

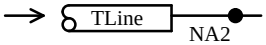
An electrical wire is a wire which will have *voltage* with respect to "ground", and through which a *current* may flow. Electrical wires are used to connect together resistors, diodes, transformers and other electrical components forming the electrical portion of the circuit to be simulated.

A "node label" may be placed on an electrical wire gives that electrical node a name. Placing node labels on two separate wires on the same page and labelling them both with the same name is an error; it does not cause the two wires to be connected.



A transmission line or cable is required to connect an electrical wire between pages.

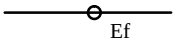
On one page

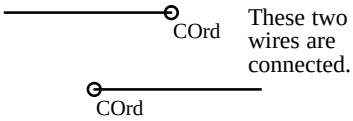


On another page

A dataflow wire is a wire which carries *data* between non–electrical components during the course of a simulation. Only a single component can be the source of data for a dataflow wire, but several components may be recipients of it. A dataflow wire has either "REAL" or "INTEGER" type associated with it, depending on what it is connected to, although conversion from one type to the other is automatically done.

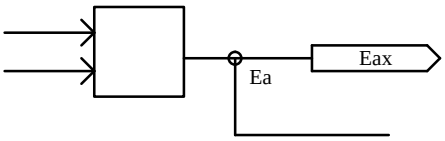
A "wire label" placed on a dataflow wire gives that wire a name. Placing wire labels on two separate wires on the same page and labelling them both with the same name logically connects the two wires.



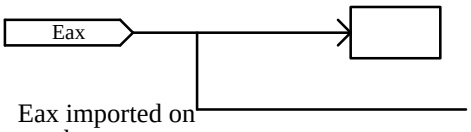


These two wires are connected.

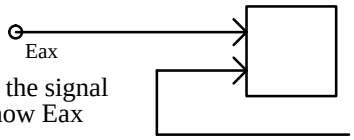
To pass a data from one page to another, it must be "exported" on the page with the component that generates the data, and imported on the pages that it is required.



Ea generated on one page, and exported as Eax



Eax imported on another page.



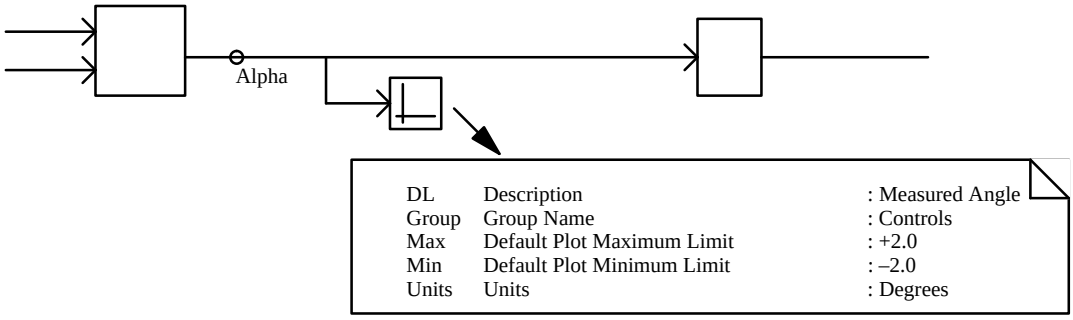
Note that the signal name is now Eax

3.2.1.8 Monitoring Quantities in RUNTIME

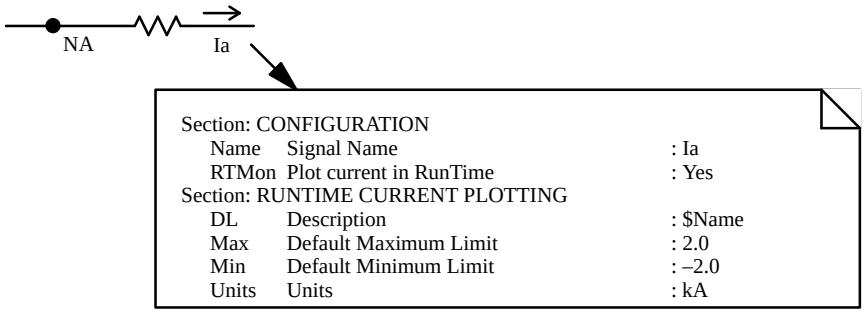
For the PSCAD simulation software to be useful, access to the information generated during the simulation must exist. For a simple circuit, it is theoretically possible to store all the simulation data for every time–step of the simulation; for more complex circuits, this is not feasible.

For these reasons, the PSCAD software requires the circuit designer to indicate the quantities which are to be monitored. These quantities are either transmitted to the RUNTIME module during the simulation, or stored in output files for later analysis, or both.

To indicate that a particular quantity is to be monitored, the circuit designer directs that signal to a "plot component". A plot component is used to assign a description and a group name to the signal, to specify the units that the signal is measured in, and to "suggest" what the range of the signal might have for the purposes of plotting.



Since the monitoring of certain quantities, such as node voltages and branch currents, occurs quite frequently, some components have the built-in capability to specify that a signal is to be monitored.



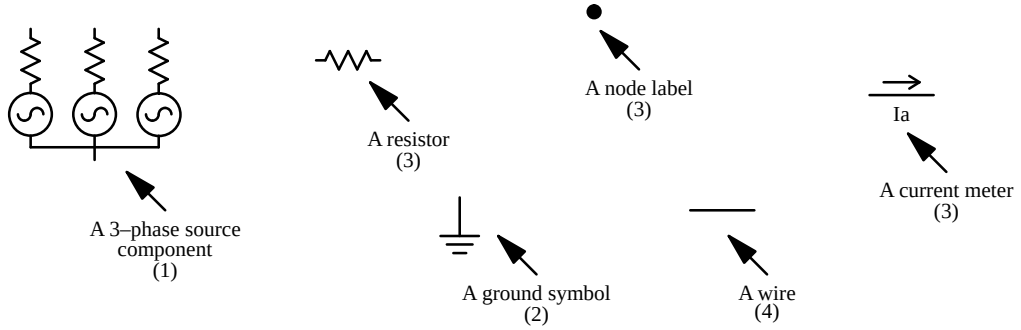
Specifying "Yes" to a "Plot _____ in RunTime" parameter causes a section to appear to contain the plotting information, and creates an information channel to RUNTIME. In these cases, the "Group" parameter is specified automatically, based on the component type, the subsystem, and possibly the component's name.

The name of the component (or any other parameter of that component) can be included in the *Description* line by using that parameter name prefixed with a dollar sign ($).

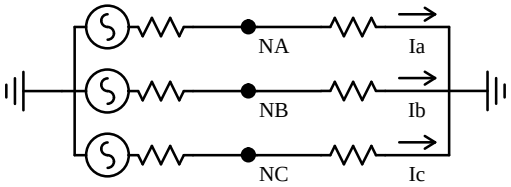
For example, in the ammeter component above, the description becomes "Ia", and the group is automatically set to "*<Subsystem Name>* | Branch Currents".

3.2.2 Sketching the Voltage Divider Circuit

The voltage divider circuit in this example uses six different components. Locate the following components in the component palette.



These components will be copied from the component palette, and placed on the drawing canvas to form the following circuit.



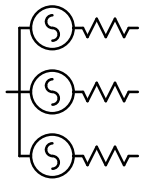
The source component will be copied first. Then, to illustrate the use of "group components," the first phase of the circuit will be created in its entirety. It will then be formed into a group component, and copied twice to form the second and third phases. These new phases will then be edited to correct phase specific data. Finally, ground symbols will be placed at each end.

The three node voltages and three branch currents will be made available for plotting in the RUNTIME module. To simplify the source model, its operational parameters will be obtained from controls in the RUNTIME module.

If the *Auto Edit* option is enabled, the corresponding edit windows will open as the components are copied <u>from the component palette to the drawing canvas</u>. If the *Auto Edit* option is not enabled, or if a component on the drawing canvas is copied, then the EDIT menu item must be selected from the component menu to change the component's parameters.

3.2.2.1 Step #1: The Three Phase Source Model

Locate the three phase source component in the drawing palette. Execute COPY from the component menu, and "drag" the copy of the component onto the drawing canvas. Place it at a location slightly left of the centre of the visible portion of the drawing canvas. This will allow the other components to be added in to the circuit without moving the source. The source parameters should be changed to the values shown here.



Section: CONFIGURATION			
Name	Source Name	: Src	
Type	Source Impedance Type	: R	
Ctrl	Source Control	: RunTime	
MVA	System Base MVA (3–phase)	: 100.0	MVA
Vm	Rated Volts (L–L, RMS)	: 230.0	kV
F	Base Frequency	: 60.0	Hz
Tc	Voltage Input Time Constant	: 0.05	sec
ZSeq	Zero Seq. Included	: No	
Imp	Impedance Data Format	: RRL Values	
Section: POSITIVE SEQUENCE RRL			
R1s	Resistance (series)	: 1.0	Ohms
Section: SOURCE INITIAL VALUES			
Es	Initial Source Mag (L–L, RMS)	: 230.0	kV
F0	Initial Frequency	: 60.0	Hz
Ph	Initial Phase	: 0.0	deg
Section: INTERNAL OUTPUT VARIABLES			
P	Name for PU Real Power (+Out)	:	
Q	Name for PU Reactive Power (+Out)	:	
V	Name for PU RMS Source Volts	:	
Theta	Name for Source Angle (degrees)	:	
Ia	Name for Phase A Current(kA)	:	
Ib	Name for Phase B Current(kA)	:	
Ic	Name for Phase C Current(kA)	:	
Section: REMOTE FAULTS			
Tf	Fault Duration	: 0.10	sec
Rf	Source Voltage During Fault	: 0.50	p.u.

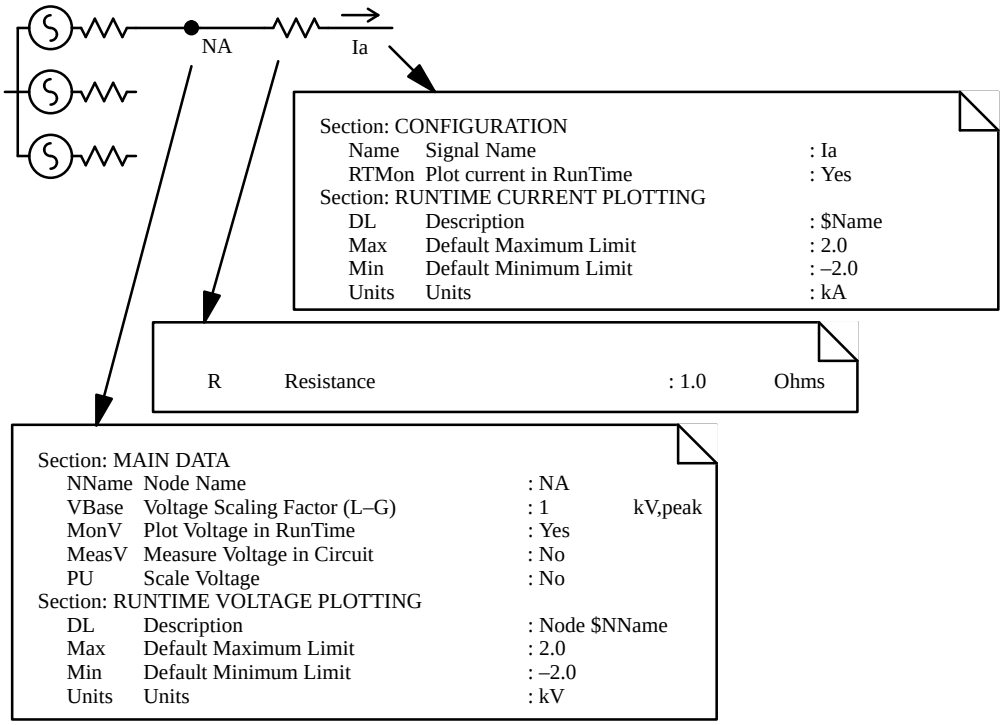
The "CONFIGURATION" section contains parameters used to describe the source. The "Name" parameter allows useful comments to be placed in the generated code, and allows the source to be referred to by name in RUNTIME.

Setting the "`Ctrl`" parameter to "RunTime" allows the creation of sliders and push-buttons to control the source in the RUNTIME module. With this set, the parameters in the "SOURCE INITIAL VALUES" section are used as initial slider positions.

Entering variable names for items within the "INTERNAL OUTPUT VARIABLES" section allows those quantities to be available to the rest of the circuit. Since our voltage divider circuit does not require this, these parameters are left blank.

3.2.2.2 Step #2: Creating the First Phase

The second step is to copy a wire, a resistor and a current meter, and attach them to the top phase of the source. Note that the wire should be stretched to a length of two dots in order for the node label to be placed at the centre. Use the parameters shown below for the corresponding components.



The EMTDC simulation software refers to nodes by numbers. The “Node Label” component creates a PARAMETER in the resulting code from the “Node Name” specified in the node label. The DRAFT software assigns a number to this parameter according to an optimal node numbering scheme, and uses the parameter to refer to the node to improve readability.

The current meter component creates a variable in the resulting code which is assigned the current which flows through the branch. The name of the variable is the name assigned to the “Signal Name” parameter.

Both the node label and current meter include the capability to monitor their voltage and current respectively. This is achieved by setting the “Plot _____ in RunTime” parameter to “Yes”. In both cases, the “Description” parameter uses a parameter reference to include the name given to that instance of the component in the description.

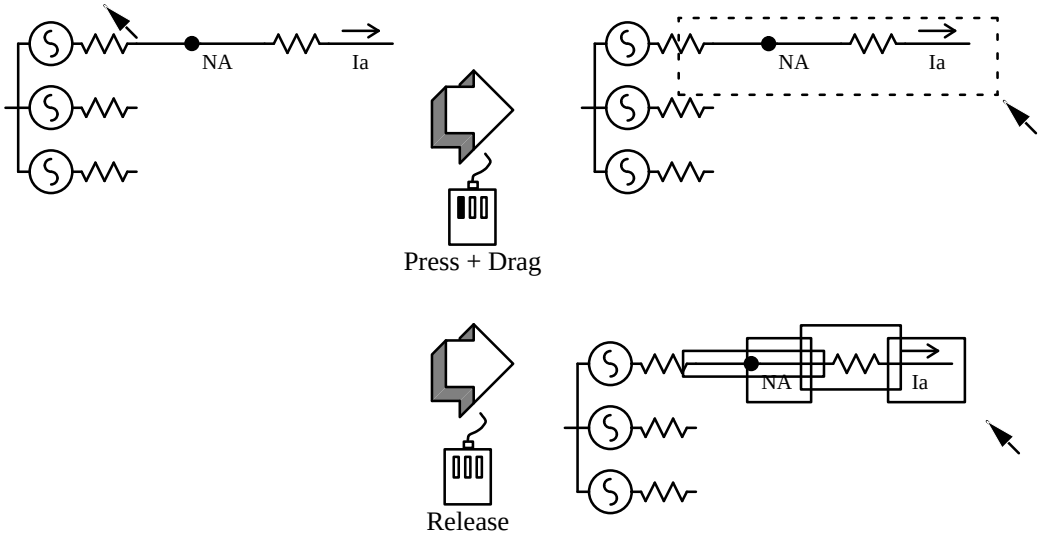
See also:

3.2.1.8 Monitoring Quantities in RUNTIME

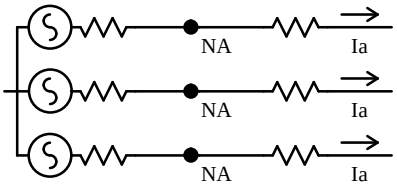
3.2.2.3 Step #3: Duplicating the First Phase

Phase A should now be formed into a “group component” and replicated twice to create the B & C phases of the circuit. First, select the phase A components:

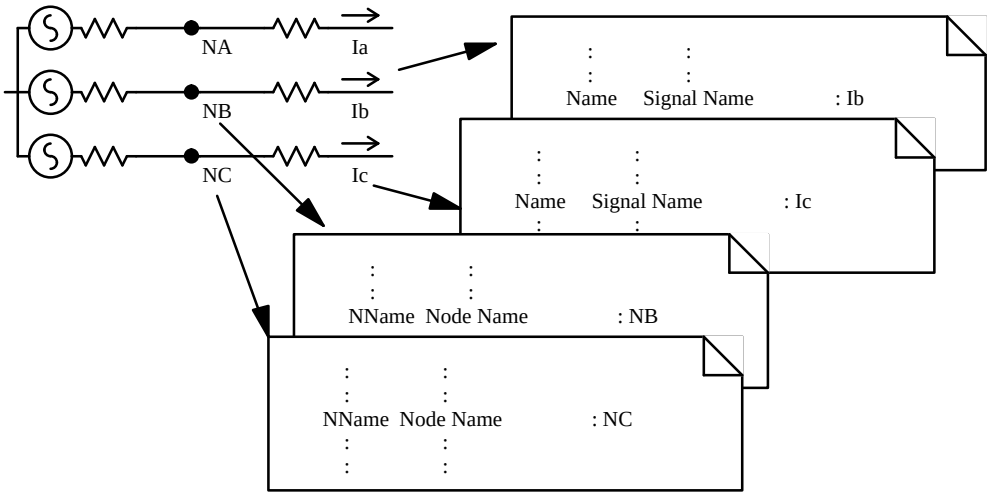
The ‘Select’ item on the component menu may also be used to select and to deselect components.



Now, select the GROUP item from the EDIT menu to form the selected components into a group component. This group may now be copied twice to create the phase B and phase C circuits.



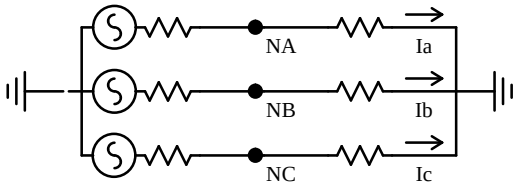
The phase B and phase C group components must now be edited to correct phase specific data, such as node names and current signal names. The group components may be degrouped first and individual components edited, or they may be edited as a complete unit.



Since the “Description” parameters contain a parameter reference to the names, and not the names themselves, changing the names will also cause the description entries to change.

3.2.2.4 Step #4: Completing the Circuit

The final step in the sketching of the voltage divider circuit is to connect both ends of the circuit to ground, as shown below.

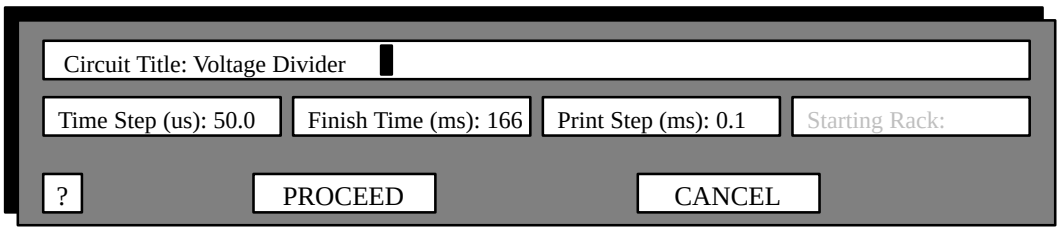


See also:

3.2.1 General Sketching Techniques

3.2.3 Entering Circuit Parameters

Like the individual components used to sketch the circuit, the entire circuit drawing has parameters which must be entered before the circuit may be compiled. These parameters are accessed by selecting the PARAMETERS option on the FILE menu. Enter the parameters shown below:



3.2.3.1 The Time–Step

EMTDC computes the *state* of the system model at every “time–step”. Smaller time–steps usually result in more accurate simulations, but increase the time the simulations take.

A value of 50µS for the time–step results in the system state being computed approximately once per degree for a 50 to 60 Hz system.

3.2.3.2 The Finish–Time

The “finish–time” corresponds to the amount of simulated time the simulation should run. Increasing the finish–time directly increases the length of the simulation.

For this case, the system *state* will be solved a total of 166mS / 50µS = 3320 times.

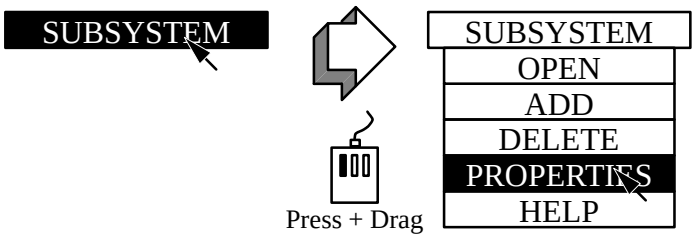
3.2.3.3 The Print–Step

The print–step allows the user to select the number of points which can be saved for plotting. In this case, a total of 166mS / 0.1mS = 1660 points can be stored.

There are many important factors which should be taken into account before deciding on the time–steps and print–steps. Consult the *EMTDC User’s Manual* for guidance in determining the values best suited for a particular case.

3.2.4 Entering the Subsytem Properties

A DRAFT circuit drawing may have many pages, also known as "subsystems". Each subsystem has associated with it a "name" and a "comment". These are entered on the "subsystem properties form", which is accessible using the "PROPERTIES" item on the "SUBSYSTEM" menu.



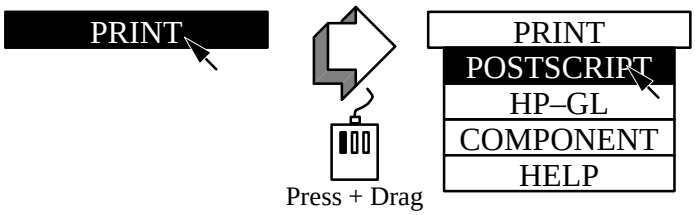
The subsystem name is used in various automatically named information channels throughout the circuit, and on printouts made of the subsystem page. The subsystem comment is used only on printouts made of the subsystem page.

If no subsystem name is specified, the name will default to "SS *<Page #>*". For instance, the first page will have the name "SS 1", the next "SS 2", and so on.

Enter "Main" for the subsystem name and then select PROCEED.

3.2.5 Printing the Circuit Drawing

If a PostScript printer is available, a printout of the circuit drawing can be obtained by selecting the "POSTSCRIPT" option of the "PRINT" menu.



This will open a form showing all the subsystem pages in the circuit drawing. In this example, only one subsystem exists. Each subsystem can be included or excluded from the print request by clicking the first mouse button on the subsystem name or icon. A subsystem which is included is surrounded by a thick border. Selecting PROCEED then completes the print request.

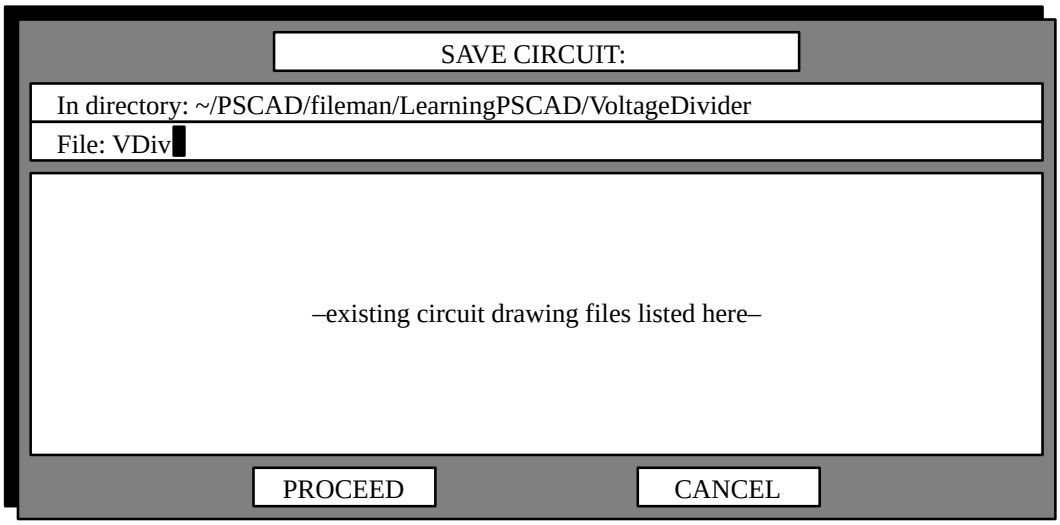
Pages can be printed out with various printing options, such as landscape or portrait page orientation, and normal or double size. Unless otherwise specified, the default is to select the "most appropriate" set of options for each subsystem page. Consult the *DRAFT Manual* for details on changing these options if required.

DRAFT can also send the circuit drawing to an HP–GL plotting device. As this is more complex than PostScript printing, consult the *DRAFT Manual* for details.

3.2.6 Saving the Circuit

"Quick" is the default action if the "Load" or "Save As" options are selected without moving to a pull–right menus.

Select FILE ⇒ SAVE AS ... menu command to save the circuit information entered during the current DRAFT session. This menu command has two sub–options: "Quick" and "Transfer". The "Quick" option, which is the default, should be selected in this case. The "Transfer" option is used to transfer DRAFT models between different sites, and is explained further in the *DRAFT Manual*



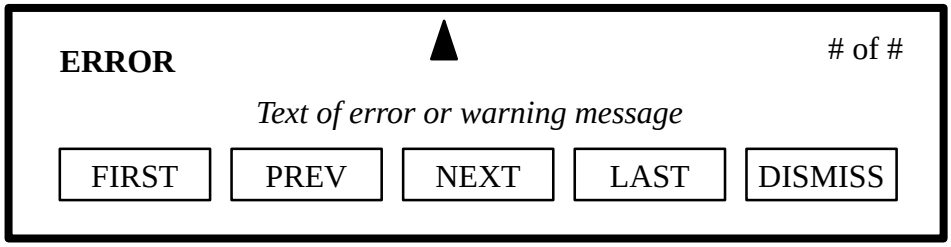
The circuit drawing must be saved before it is compiled, in order to give the circuit drawing a name. This name is used as a filename prefix for the files generated by DRAFT.

3.2.7 Compiling the Circuit

To compile the circuit, select the COMPILE option located near the top center of the DRAFT window. This procedure will produce all files necessary to run the simulation.

If the circuit has been properly entered, the text "*Compile complete. 0 error(s) 0 warning(s)*" will appear in the message box along the bottom edge of the DRAFT window.

If any errors or warnings are generated during the compile, a locater window will appear at the location of the problem.

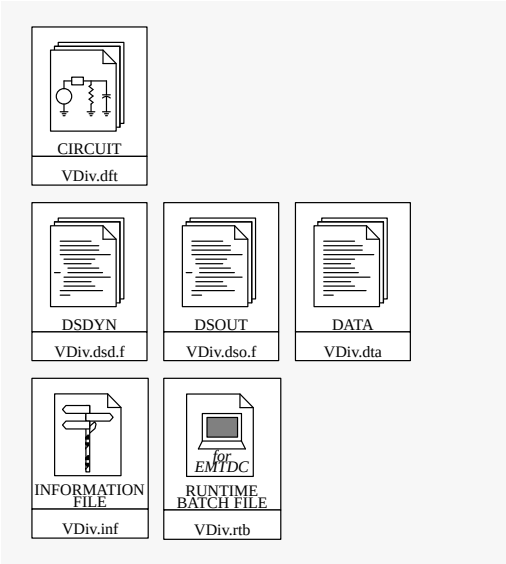


Ensure that the circuit drawing and parameters are correct, and recompile the circuit. Once the circuit compiles without problems, save the circuit drawing.

3.2.8 Iconifying DRAFT

The user is now ready to perform the simulation of the voltage divider circuit. The DRAFT module should be temporarily iconified back into the FILEMANAGER by selecting the ICON option from the top menu.

Notice that there are a number of new file icons which have appeared in the user's work area in the FILEMANAGER window. The new file icons are bounded with dotted lines since they have been automatically placed. The user may choose to move these icons to their permanent position.



.dft file contains the DRAFT circuit data

.dsd: The EMTDC DSDYN file (Fortran statements)

.dso: The EMTDC DSOUT file (Fortran statements)

.dta: The EMTDC DATA file (EMTDC Format)

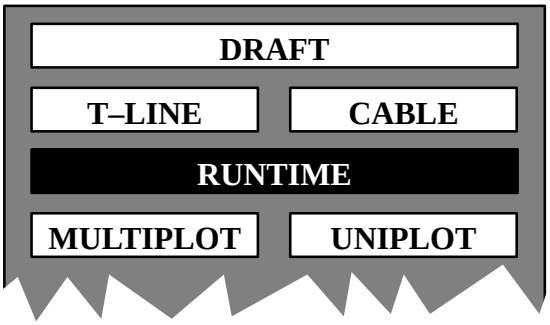
.inf: Information file used by the RUNTIME software

.rtb: Batch file used by the RUNTIME software

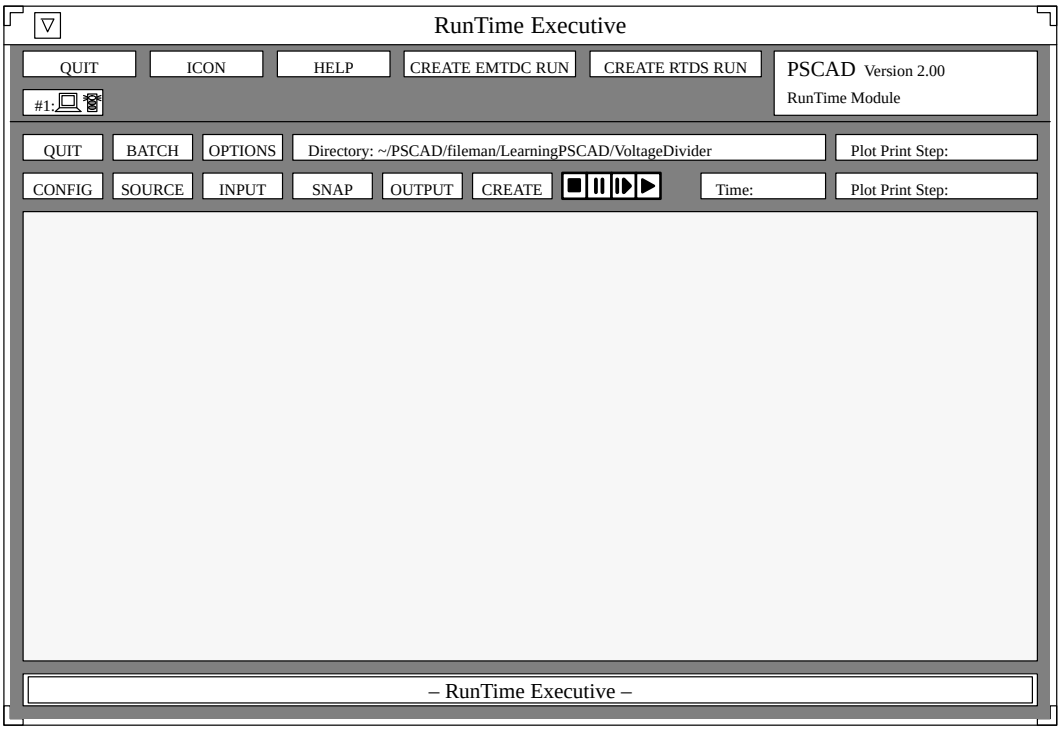
3.3 PERFORMING THE SIMULATION USING RUNTIME

The PSCAD/RUNTIME module allows the user to create one or more OPERATOR'S CONSOLE windows. Each window is dedicated to a particular EMTDC run. If the user's workstation is linked to a local area network, other computer workstations accessible from the network may be used to perform the numerically intensive simulations. Data produced by the simulations is reported back and displayed on the local workstation.

To start the RUNTIME module select its box from the right hand side of the FILEMANAGER window.



When RUNTIME is started, a single EMTDC OPERATOR'S CONSOLE window is automatically created and displayed.



3.3.1 Setting up the EMTDC OPERATOR'S CONSOLE

Several items of information are required in order to simulate a power system circuit from the EMTDC OPERATOR'S CONSOLE:

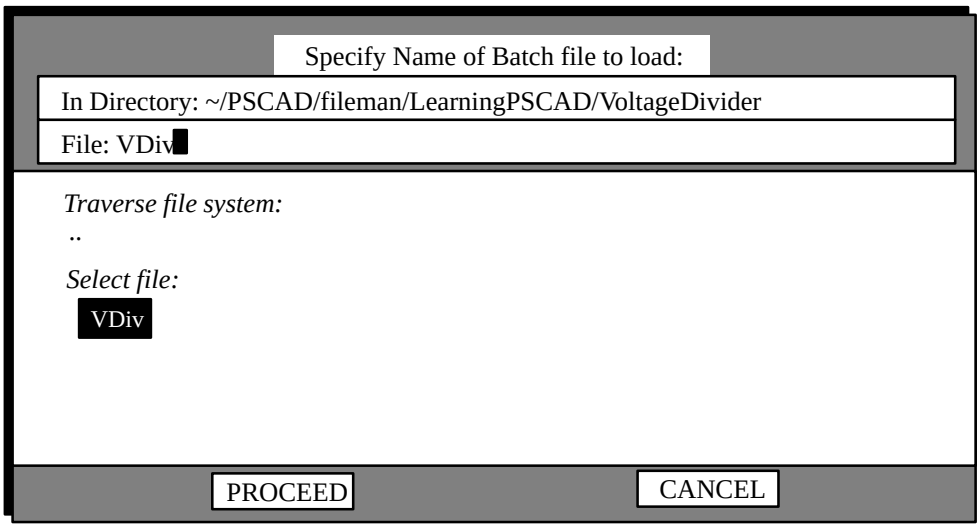
*These items are entered in the **CONFIG**, **SOURCE**, **INPUT**, **SNAP** and **OUTPUT** sections.*

- the EMTDC library dimension version to use
- the Fortran dynamics and output subroutine filenames, as well as any other subroutine files to be included
- the information file and the starting data or snapshot file
- whether or not to write snapshot files, and the time of these snapshots
- whether or not to write output files, and the name to use for these files

Since these items are always required, the DRAFT module creates a RUNTIME batch file with "default" information when a case is first compiled. When a case is recompiled, DRAFT will only modify the subroutine file information of an existing batch file.

To load the batch file associated with the voltage divider case, select the LOAD option from the BATCH menu.

This will open a "File Requestor" form, where the name of the batch file to load may be entered. Either type "VDiv" in the "File:" input box, or click on the "VDiv" filename in the "*Select file:*" section. After specifying the batch file to load, select PROCEED.



When the form vanishes, the OPERATOR'S CONSOLE will return to its original appearance. To view the effect of loading the batch file, select SETUP from the CONFIG, SOURCE, INPUT, SNAP, and OUTPUT menus.

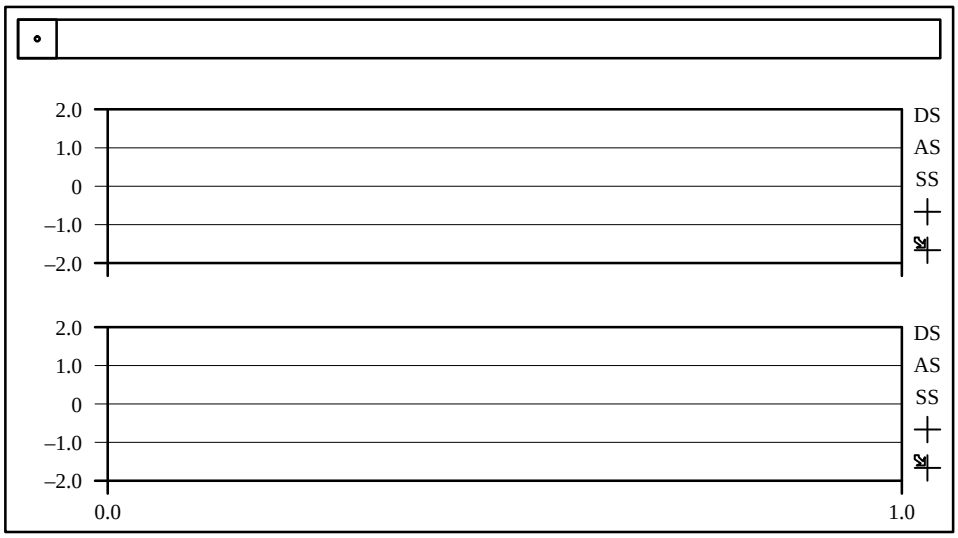
3.3.2 Creating Monitoring Interface Objects

As an EMTDC simulation runs, the information generated by the simulation can be recorded in an “output file” at a rate specified by “Output Print Step”. This output file is available for examination, after the simulation is complete, to see what occurred during the simulation.

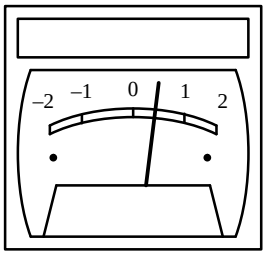
The information generated by the simulation cab also be directed to the OPERATOR’S CONSOLE, for display in one or more monitoring interface objects. The rate at which this information is sent is specified by the “Plot Print Step”.

There are two different monitoring interface objects which can be created for use on the OPERATOR’S CONSOLE window.

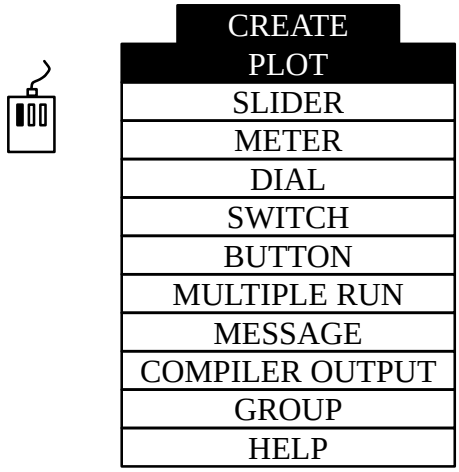
The first monitoring interface object is the “plot” component, which can display upto five (5) graphs with up to three (3) information channels per graph. The signals are plotted against time from the start of the simulation. The plot component is most suited for displaying information which varies quickly with respect to time (eg, AC system quantities), and information for which a record over the entire run is desired.



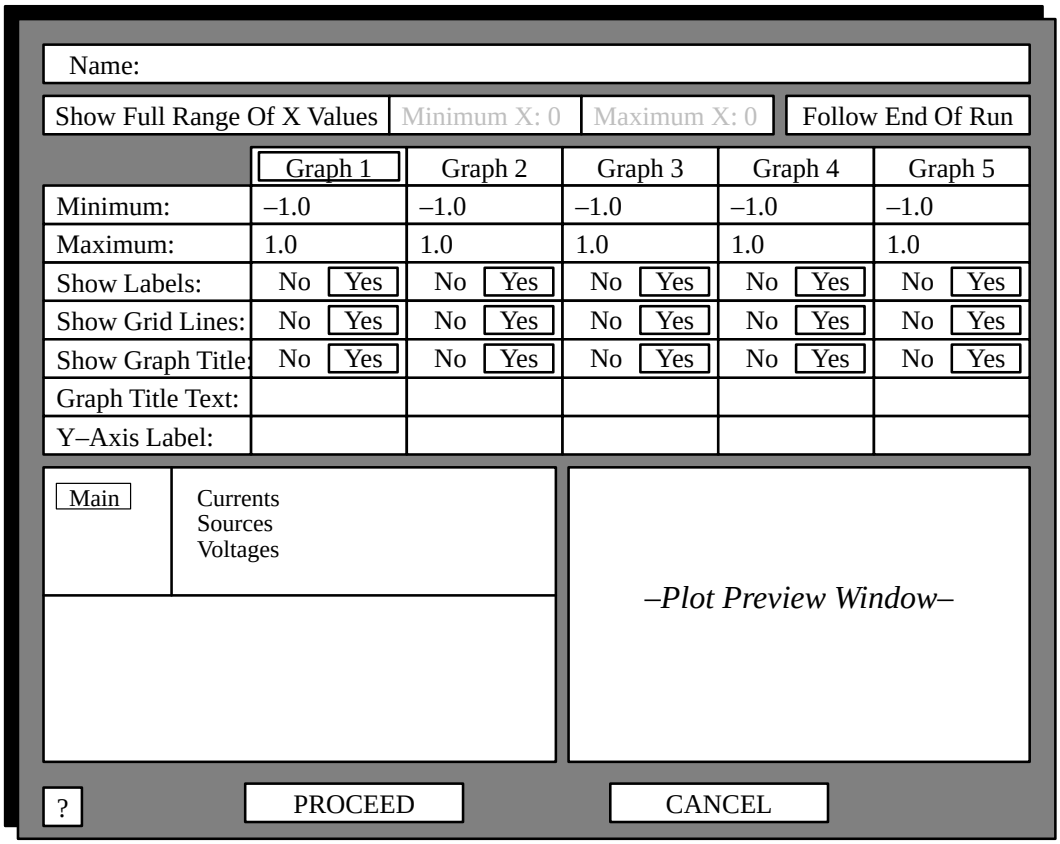
The second monitoring interface object is the “meter” component, which displays the instantaneous value of one information channel. The meter component is more suited to displaying slowly varying (or infrequently changing) quantities.



In this voltage divider example, three node voltages and three branch currents were identified as output variables. Since these are AC quantities, the plot component is more suitable for displaying these quantities. Select PLOT from the CREATE pull–down menu.



This will open the plot component's "Properties Form", where the appearance of the plot may be control from, and the quantities to be plotted may be selected.

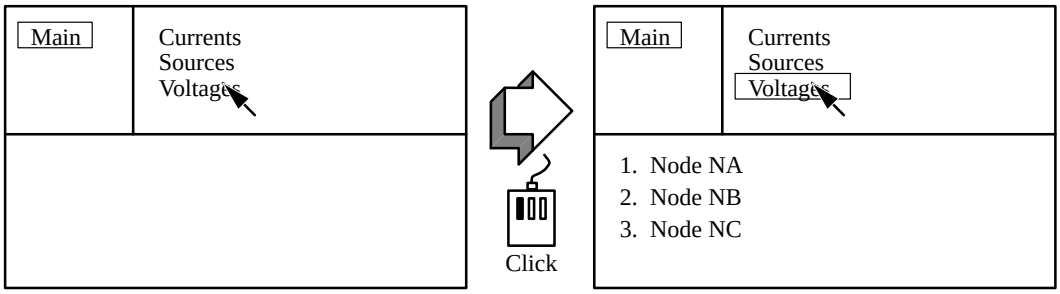


The upper half of this form controls the appearance of the graphs on the plot component. The lower left area is an *information channel selection area*, used to select the quantities which are graphed. The lower right are shows a miniaturized picture of how the plot component would look if PROCEED was selected at that point.

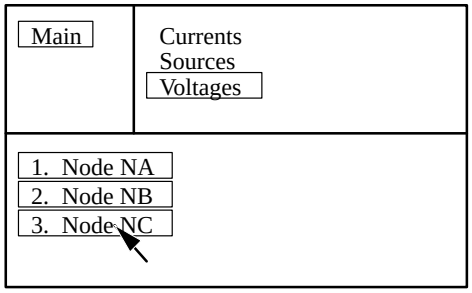
The information channel selection area is composed of two parts: a group selection area at the top, and a channel description selection area at the bottom. The group selection area is hierarchical in nature. Selecting a group will cause any sub–groups within that group to be displayed to the left of a vertical line. These sub–groups may in turn have sub–groups. As different groups are selected from the top area, the bottom area changes to reflect the channels which are in that group.

Selecting a different group from the top area does not affect which information channels are currently selected. As such, it is possible to display on the same graph quantities which are in different groups.

Choosing the “Voltage” sub–group will display the channels which are part of the “Main : Voltages” group.

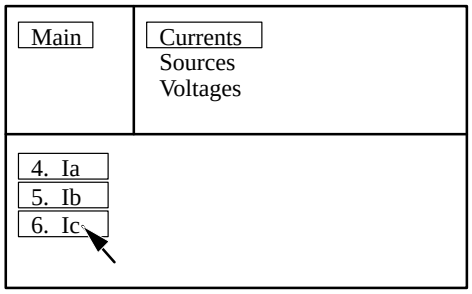


Select the “Node NA”, “Node NB” and “Node NC” channels under this group to add them to the first graph.



Note that as the information channels were selected, the “Minimum:”, “Maximum:”, and “Y–Axis Label:” parameters for the first graph were changed to the values which were specified in DRAFT for those information channels. The plot preview area also changed to reflect how the plot component currently looks.

Select the “Graph 2” graph selector, and then add the current information channels from the “Main : Currents” group. Again, note the changes to the plot parameters and the plot preview area.

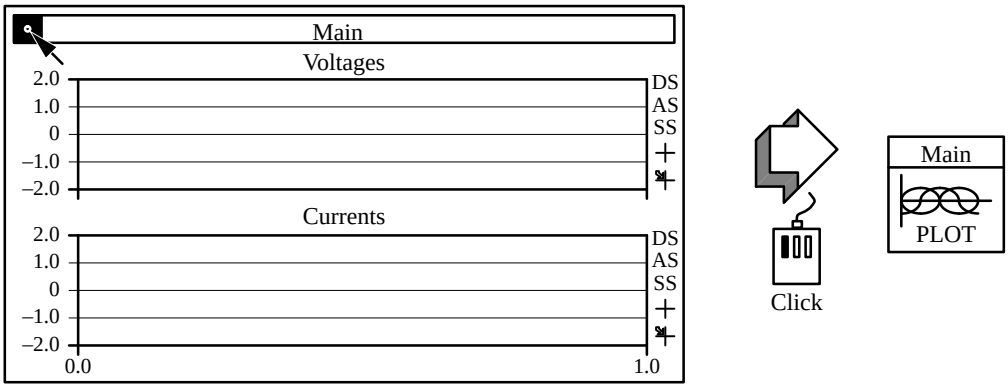


Select PROCEED to complete the plot component creation request.

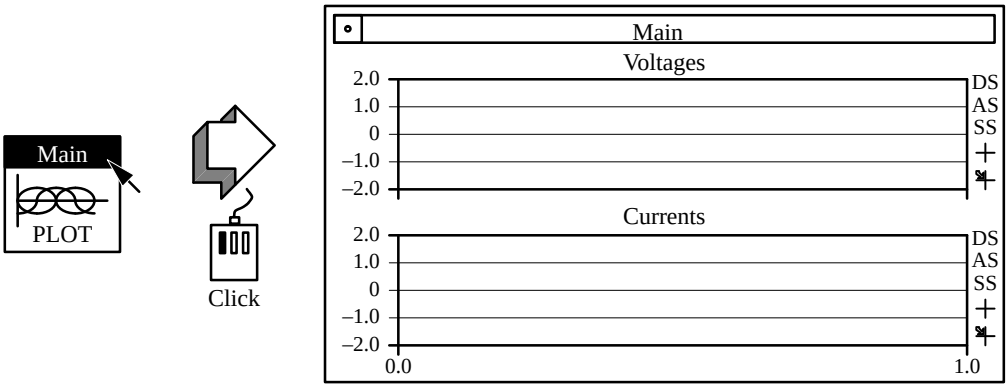
Once PROCEED has been selected, a plot component with two graphs appears on the OPERATOR'S CONSOLE. Since no name was specified on the properties form for the plot component, an automatic naming feature is used. The plot component name is determined from the group names of all channels which appears on the plot. Since all of the group names have "Main" as a common parent, the component name is set to "Main". Similarly, the graph titles become "Voltages" and "Currents"

To move the plot, press and hold down the left mouse button over the plot window, and "drag" it. To change the plot's size, press and hold down the left mouse button over the bottom–left or bottom–right corner of the plot's window (the cursor changes shape), and "drag" this corner to alter the plot's size.

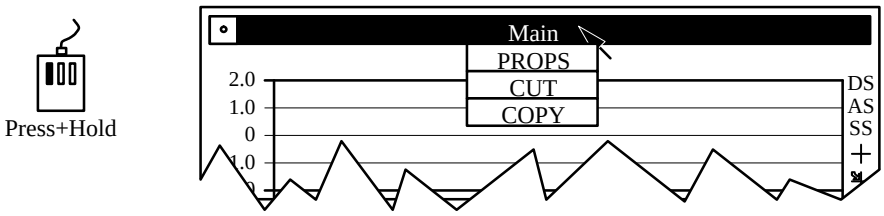
The plot component may be changed into an icon, making more viewing room for other components. This is accomplished by clicking the first mouse button on the icon box at the top left of the components.



To move the plot icon, press and hold the left mouse button over the lower part of the plot icon and drag it to the desired position. To uniconify the plot component, click the first mouse button on the upper part of the plot icon.



The properties of the component may be revised, or the plot itself may be cut using the plot components pull–down menu.



3.3.3 Creating Control Interface Objects

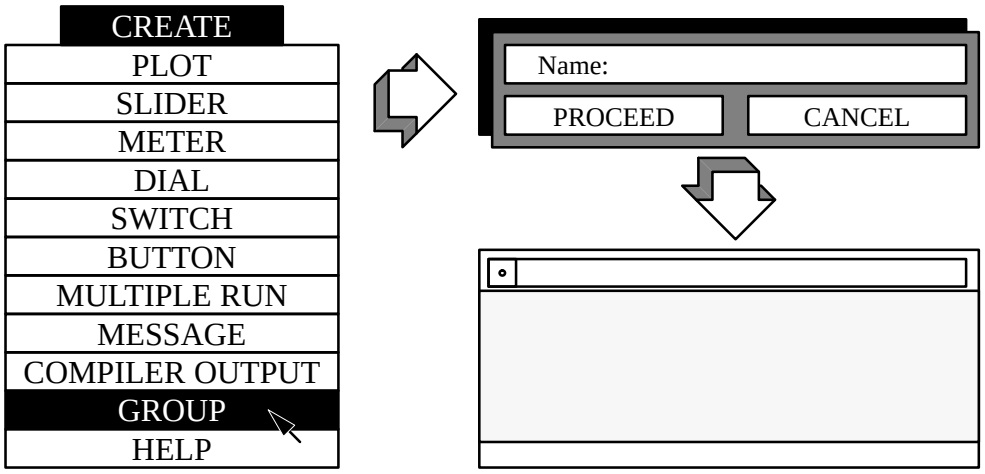
Several different control interface objects can be created for use on the OPERATOR'S CONSOLE window. The source component in this voltage divider example uses two types of control objects: "sliders" and "buttons".

Plots and Meters may also be created inside group components.

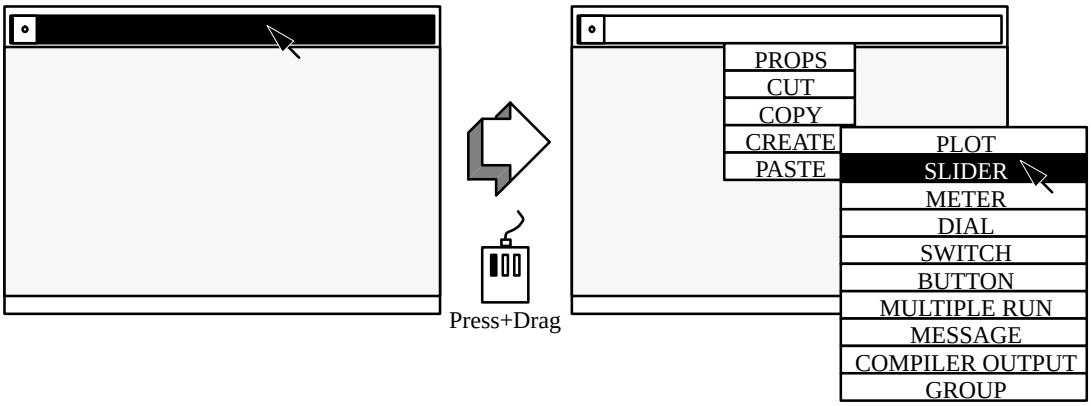
Although the control interface objects can be created directly on the OPERATOR'S CONSOLE window, it is often useful to create them inside a "group component". There are several reasons for this.

- Control interface objects do not have the capability to be reduced to an icon. A group component which contains other interface objects can be reduced to an icon.
- The control interface objects are generally small, therefore the size of the title bar of the component does not leave room for long descriptions. For instance, "Subsystem #1 : Sources : SRC1 : Magnitude" could be shortened to merely "Magnitude". However, this may cause confusion as to which magnitude the control refers to. This confusion could be eliminated by assigning a name of "Subsystem #1 : Sources : SRC1" to the group component in which the control was placed.
- Groups may be created inside other groups, allowing for very large hierarchies of interface objects. This is especially recommended for larger simulations.

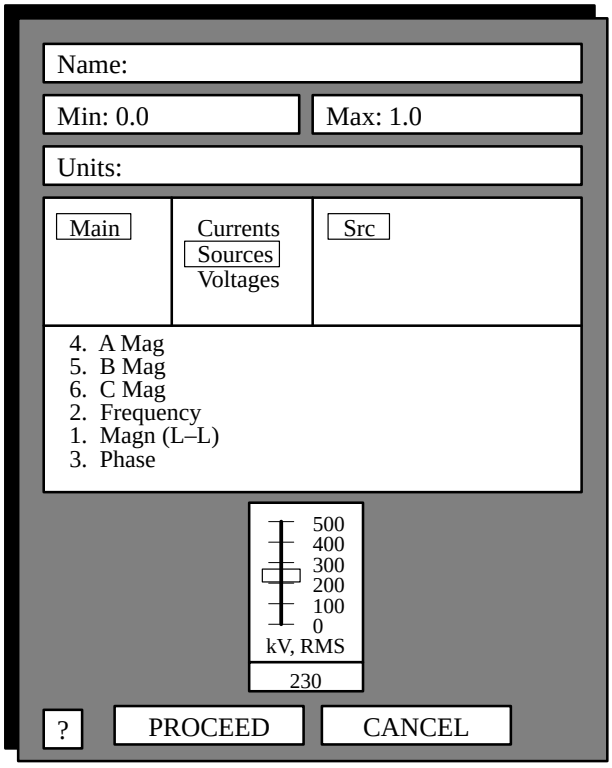
To create a group component, select GROUP from the CREATE pull–down menu. Leaving the name field blank enables an automatic naming feature for the component group. The common group prefix of all items placed in the group will become the name of the group. Once PROCEED is selected, the group component will appear on the OPERATOR'S CONSOLE window. The group component may be moved, iconified and resized in exactly the same way as the plot component.



Interface objects can be placed in the group component in one of two ways. If the interface object already exists on another part of the OPERATOR'S CONSOLE or in another group component, that component may be copied or cut to the "clipboard" and then pasted in the desired group using the PASTE item from the group component's pull–down menu. To create a new interface object inside the group component, the CREATE pull–right menu of the group component's menu is used.

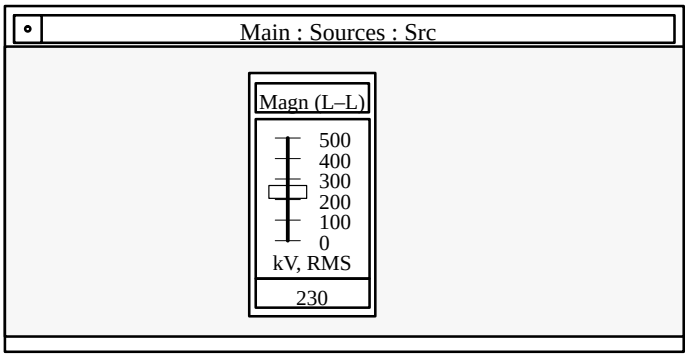


Selecting SLIDER from the CREATE pull–right menu opens the slider component property form, where the information about a slider may be entered. Notice that the information channel selection area on the form immediately displays the "Main : Sources : Src" group, as that is the only group containing slider channels.



Select the "Magn (L–L)" control channel. Note that the "Min:", "Max:" and "Units:" parameters change to values appropriate for a slider controlling the magnitude of a voltage source.

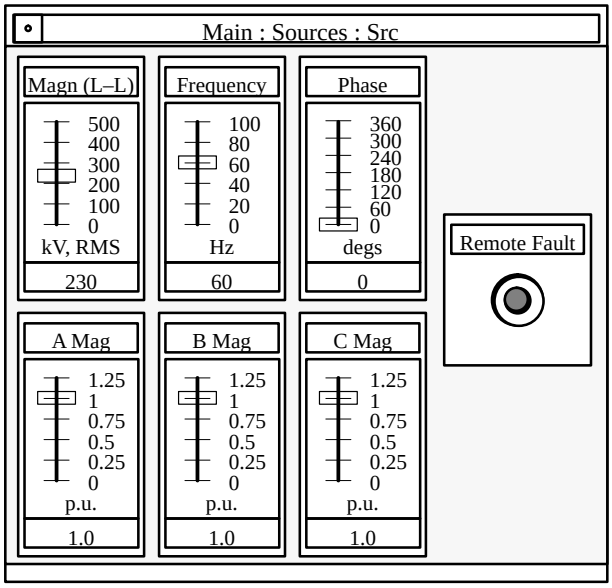
Select PROCEED to complete the slider creation request. Notice that the name of the slider is set to “Magn (L–L)” and that the name of the group containing the slider is set to “Main : Sources : Src”.



The slider may be positioned anywhere inside the group by dragging the slider with the first mouse button. To move the slider outside the group, it must be CUT from the group, and then PASTE’d elsewhere.

Position the slider in the upper left corner of the group, and then create the other five source sliders inside the group.

Finally, create a BUTTON control interface object inside the group component for the “Main : Source : Src : Remote Fault” control channel. When complete, the group will look similar to the following:

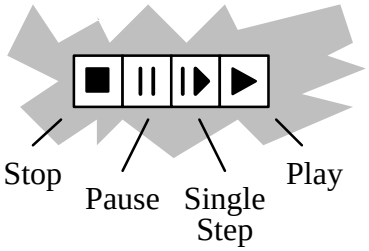


This completes the creation of the interface objects for the voltage divider case.

If all the control interface objects for a particular simulation case are not created in the OPERATOR’S CONSOLE, the channels assume the default values that the control objects would be created with if they were created.

3.3.4 Starting the Simulation

Towards the top of the OPERATOR'S CONSOLE window is a set of controls similar in marking to controls which appear on tape records and VCR's. These controls are used to start, pause, single step, and stop the EMTDC simulation.



To begin the simulation, select the "Play" box from the top of the OPERATOR'S CONSOLE window. The following messages will sequentially appear in sequence in the message box at the bottom of the OPERATOR'S CONSOLE.

Preparing temporary work area ...

Preparing temporary work area ... Compiling (2 files) and linking ...

Preparing temporary work area ... Compiling (2 files) and linking ... Starting Run ...

Preparing temporary work area ... Compiling (2 files) and linking ... Starting Run ... Run Started.

"Preparing the temporary work area" entails creating a "temp" directory and copying the files required for the EMTDC simulation to this directory.

"Compiling (?? files) and linking" is the phase where RUNTIME starts the Fortran compiler to compile the DSDYN and DSOUT subroutines, and link these to the appropriate EMTDC library.

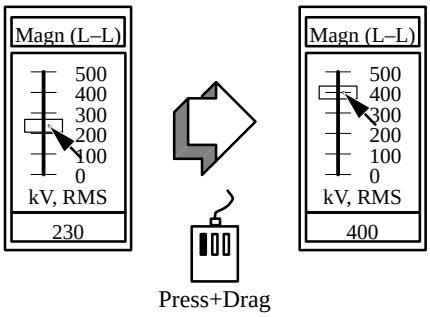
"Starting Run" is the phase where the resulting simulation executable is started and communication with the OPERATOR'S CONSOLE is established.

"Run Started" signals the end of the above step, indicating that the simulation has begun.

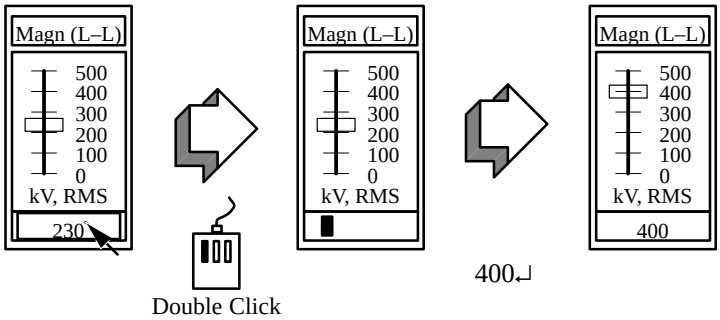
If any errors occur during this sequence, a "Compiler Output" window opens with information regarding the problem which has occurred. The shell from which PSCAD was started, or the console window, may also display diagnostic messages which may help indicate where the problem has occurred. If any problems are encountered, please consult the appropriate manual section, or the system administrator.

3.3.5 Interacting with the Simulation

The OPERATOR'S CONSOLE interface objects such as sliders and push-buttons allow system parameters to change interactively while the simulation in progress. In this voltage divider case, six sliders allow control of various parameters of the source model. A slider value can be changed by pressing the first mouse button on "slide bar" of the slider, and dragging it up or down on the slider.



The slider value can also be changed by activating the input box at the bottom of the slider, and entering the desired value.



The change in slider value is immediately reflected by a corresponding change in the signals on the plot component.

The "Remote Fault" push–button component can be used to simulate a remote fault, in which the source's magnitude is reduced for a short period of time.

To change several controls at once, the operator may select the "Pause" control at the top of the OPERATOR'S CONSOLE, make the desired changes, and then "Un–pause" the run by re–selecting the "Pause" control.

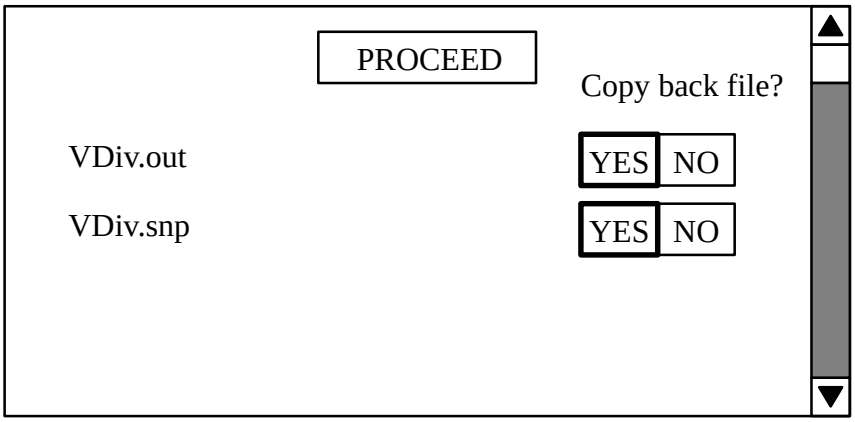
During the course of the run, the curves on the two graphs of the plot component may obtain values above or below their respective graphs. The graphs can be rescaled to display the entire vertical range by selecting the "AS" or "SS" controls at the right side of the graphs. The AS control will "Auto Scale" scale the graph, where as the SS control will "Smart Scale" the graph. The smart scale option will choose grid limits which are "nicer" than the auto–scaled limits, but results in more "white space" on the graphs.

The use of sliders, pushbuttons, dials, and switches to control the simulation will not result in repeatable simulations, as the simulation time at which these controls are modified may vary.

3.3.6 Ending the Simulation

The EMTDC simulation can be stopped at any time by selecting the “Stop” control at the top of the OPERATOR’S CONSOLE. If the not manually stopped, it will will automatically stop when the simulation time reaches the *finish time* that was specified for the case.

The “Copy Back” form will open when the run is stopped if output files were being created, or if any snap–shots were taken during the simulation, . This form allows the operator to control whether the output file(s) from the latest simulation should be “saved” (copied from the run directory to the current directory) depending on whether the simulation was “interesting”.



Once PROCEED is selected, the desired files are copied back, and the operator can re–run the simulation by selecting “Play” from the top of the OPERATOR’S CONSOLE.

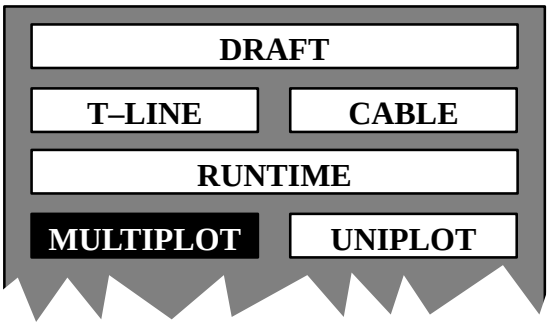
After re–running the simulation, a warning message may appear for various files in the “Copy Back” form, warning the operator that files by these names exist in the current directory, and that these will be replaced by the new files if “YES” is selected. Generally, it is acceptable to overwrite the older files, but if not, the RUNTIME module may be reduced to an icon and the old files renamed to prevent being overwritten.

Re–run the simulation, changing slider values and causing remote faults, to create an “interesting” output file to use in the next step of this tutorial. Copy the output files back, and then select ICON from the top of the RUNTIME module to reduce it to an icon inside the FILEMANAGER.

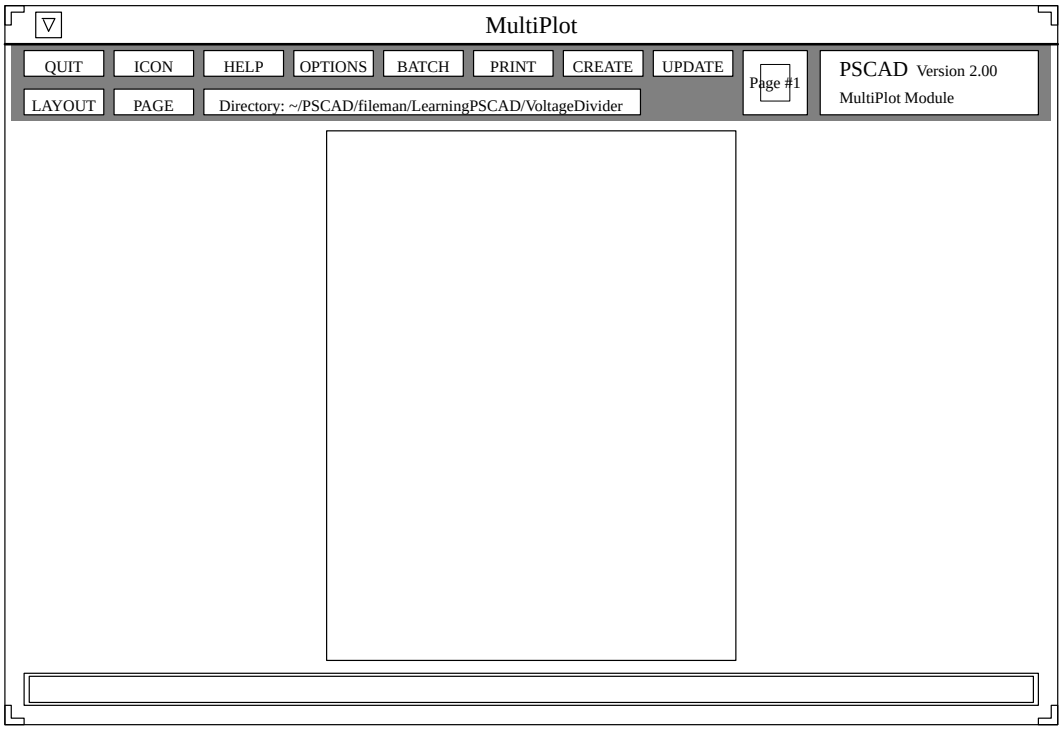
3.4 PLOTTING AND ANALYZING THE RESULTS USING MULTIPLOT

The PSCAD MULTIPLOT module enables the user to create report quality printouts of the data produced by the EMTDC simulations. In addition, the format of the pages can be stored and subsequently re–used to quickly produce several sets of printouts of similar data.

In order to start the MULTIPLOT module select its box from the right hand side of the FILEMANAGER window.



It may take a few seconds before a new window, similar in appearance to the one shown below is displayed on the workstation's monitor.

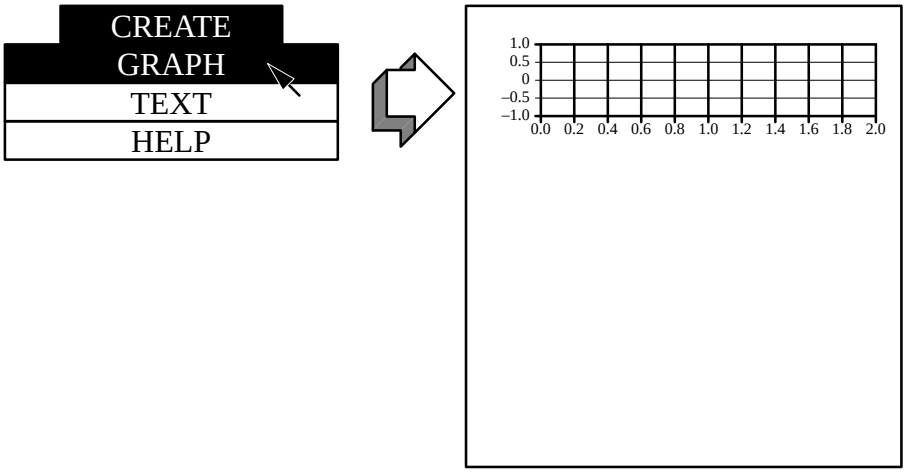


3.4.1 Adding Graphs to a Page

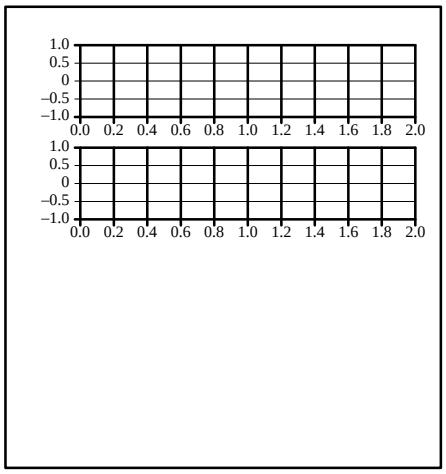
3.4.1.1 Creating Graphs

The MULTIPLOT module is based on the concept of pages. When MULTIPLOT is initially started, only one page is present. Additional pages can be added by selecting "ADD" from the "PAGE" menu. Each page is initially empty when created.

To plot the channel information generated from the EMTDC simulation, graphs must be added to the page. Select "GRAPH" from the "CREATE" menu.



This graph will be used to plot the node voltages. Create a second graph for the branch currents in the same fashion. Notice how the graphs are automatically positioned on the page.

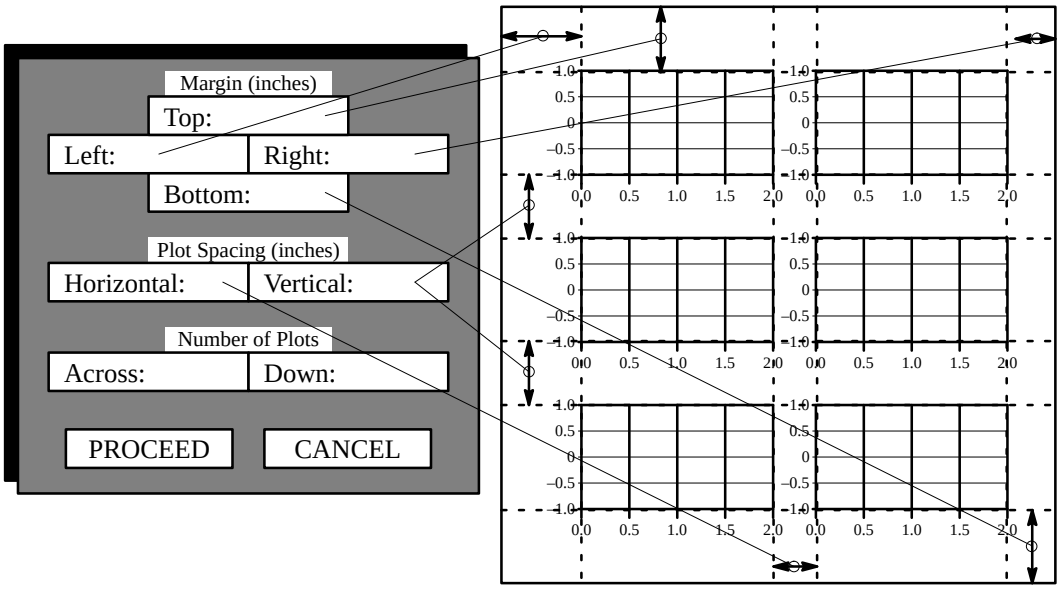


MULTIPLOT will use a default layout of one graph horizontally across the page and four graphs vertically spaced across the page. When a new page are added, the default layout for new page is set to the layout of graphs on the current page.

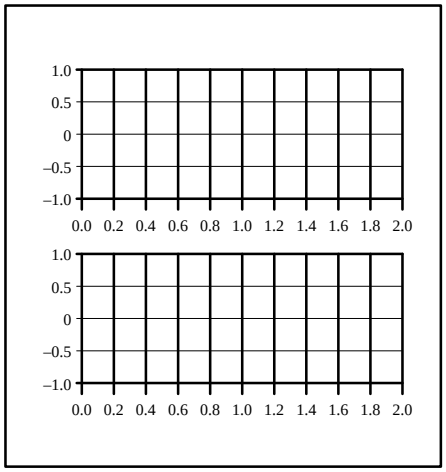
3.4.1.2 Altering the Page Layout

Since only two graphs will be present on this page, the page layout must be adjusted. Select “GRIDDED” from the “LAYOUT” menu to open the “Layout Form”.

The page layout is based on a regular arrangement of graphs, page margins, and inter-plot spacing. Note that the margins and inter–plot spacing do not take into account any text around the graph, only the grid extents. If an irregular arrangement of plots is required, the positions and size of the plots must be changed manually.

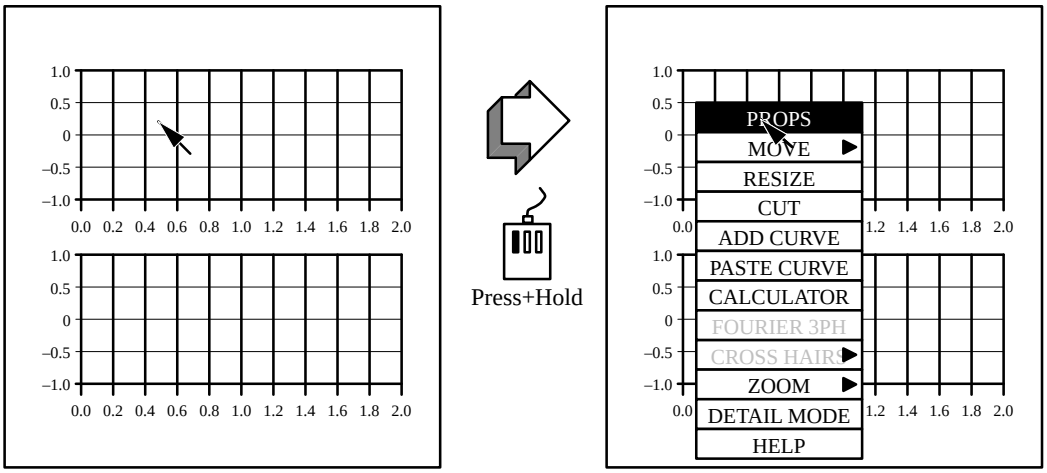


Specify 1.5 inches for the top, left and bottom page margins, and 1 inch for the right page margin, to leave room for hole punching in the left margin and text in the top and bottom margins. Specify a vertical plot spacing of 1 inch to leave a gap between the graphs. Since there will only be 1 graph across the page, there is no need to change the horizontal plot spacing. Finally, set the number of plots across to 1, and the number of plots down to 2. Select PROCEED to apply the layout to the page.

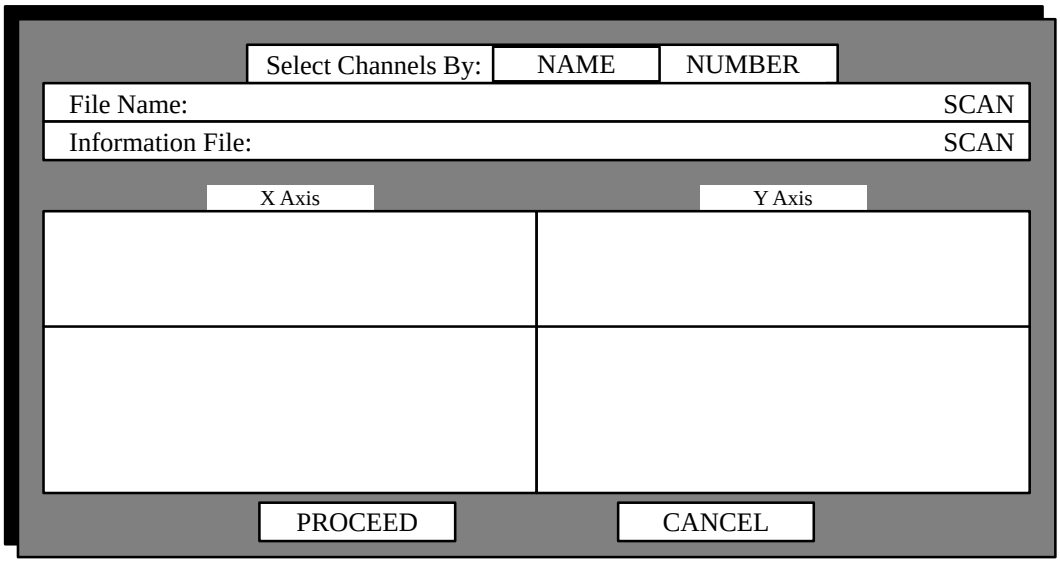


3.4.1.3 Adding Curves to a Graph

In order to add a curve to a graph, MULTIPLOT must know which graph the curve is being added to. Using a menu from the top of the module would not provide this information. Instead, each graph has its own menu. To activate this menu, move the cursor inside the grid of the graph, and press and hold the first mouse button.



Selecting “ADD CURVE” from the graph menu opens the following form:

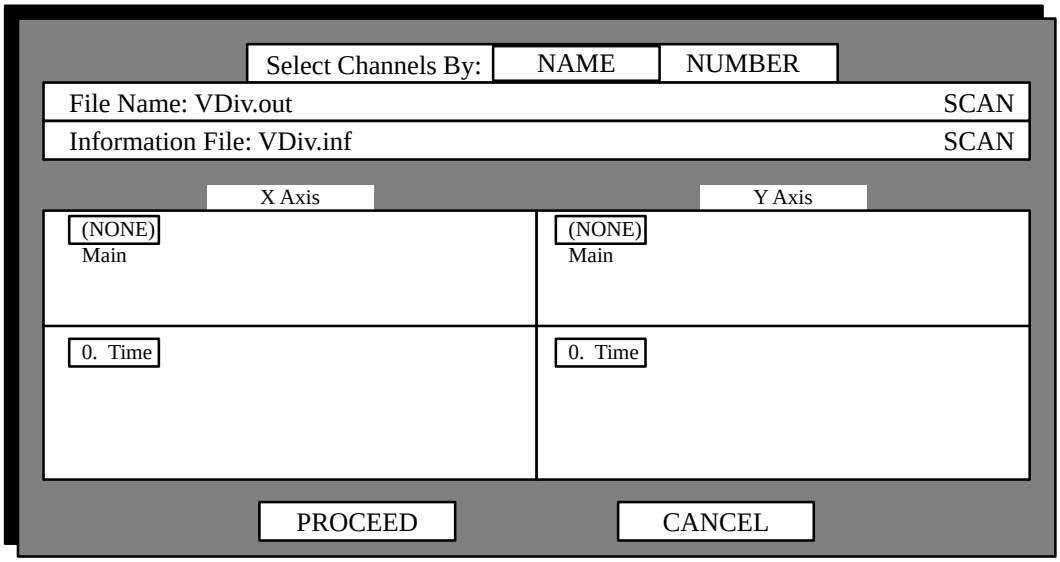


Channel information is stored in a collection of datafiles. Up to 10 channels, as well as the time information channel, are stored in each datafile. The first 10 channels are stored in a file named “<*filename*>–1”, the next 10 in “<*filename*>–2”, and so on.

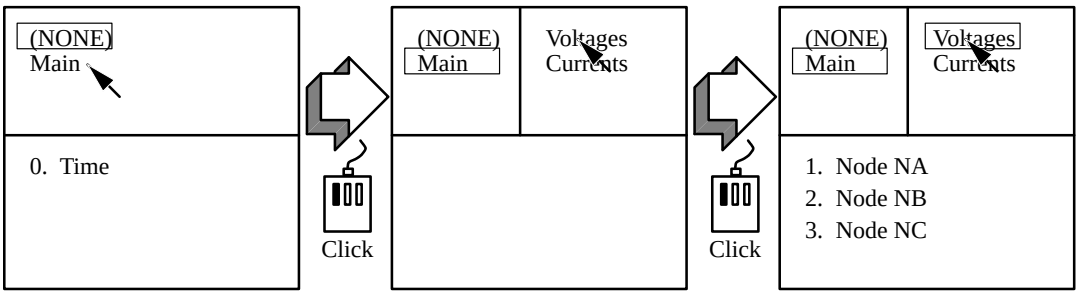
When “Select Channels By: Number” is specified, the full file name “<*filename*>–#” and the channel number within that file are required. For instance, the 37th channel of “garb.out” would be channel #7 of “garb.out–4”.

When “Select Channels By: Number” is specified, the “<*filename*>” prefix. the name of the information file (generally “<*filename*>.inf”), and the channel name are required.

Enter “VDiv.out” at the “File Name:” prompt and “VDiv.inf” at the “Information File:” prompt, or select “SCAN” in each box and select the appropriate file from the resulting file requestors. In either case, the channel selection area at the bottom of the form changes as follow:



Under the “Y Axis” heading, select the “Main” group. A vertical bar appears and the sub–groups of “Main” appear to the left of this. Select the “Voltages” sub–group. The voltages channels appear in the channel selection area.



Select “Node NA”, and then choose PROCEED to load the curve information into the graph. The process may then be repeated for the other two voltage channels. Note that the old information remains in the form when it is re–opened, thus simplifying the procedure.

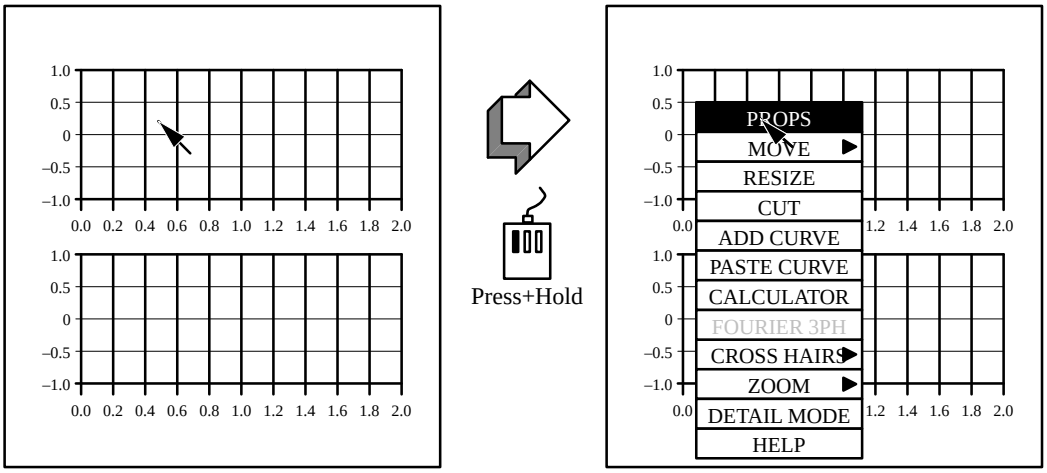
Once all the voltages have been added to the first graph, the entire process can be repeated for the second graph to add the current information.

When adding curves to a graph, ensure that the “Y Axis” channel is set to the desired channel, and the “X Axis” channel remains set on “Time”. Reversing the order will cause the graph to display time along the Y–Axis! Although it will not create any technical problems, thus results seem somewhat unnatural.

X verses Y plotting is achieved by selecting an “X Axis” channel other than time.

3.4.1.4 Changing the Graph Properties

The X–axis label, Y–axis label, and Title are properties of a graph. To change these, select the “PROPS” item from the top graph’s graph menu.



This opens the properties form for the top graph. For the graph title parameter, enter “Voltages”; for the vertical axis label, enter “kV”. Since the horizontal axis for the top graph is the same as for the bottom graph, “HIDE” may be selected for the horizontal axis label.

The length of the names for the curves added to the graph and the size of the graph will determine whether the curve labels overlap one another. The parameter “Maximum Number of Curve Labels in a Row” controls the number of curve labels placed in a row at the top of the graph. Since the labels in this example are fairly short, this parameter can be safely changed to three (3).

The remaining parameters affect the way numbers are displayed, whether the limits are rounded to “nice” values, the number of grid lines, and the style used for labelling curves. There are also parameters for controlling the limits of the information displayed, but these are more easily controlled using the “DETAIL MODE” item of the graph menu.

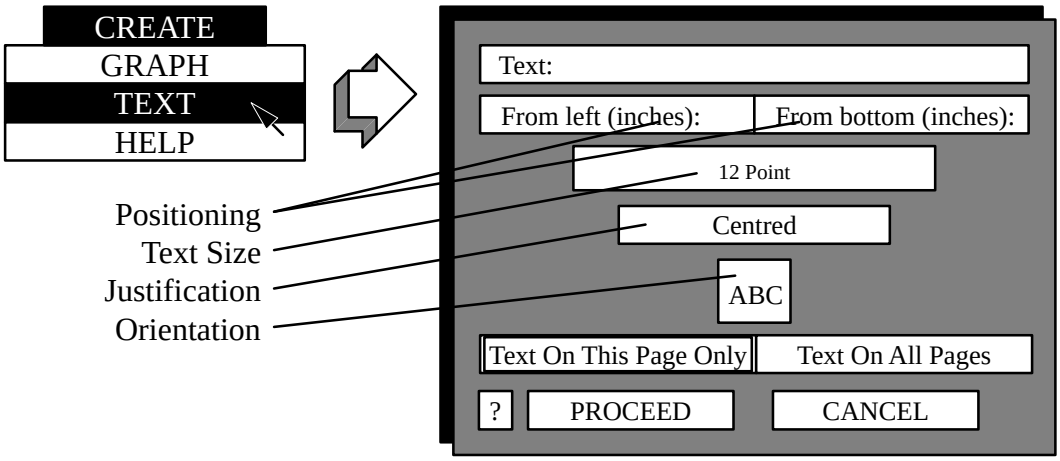
Select PROCEED to complete the properties modification.

Open the properties form for the second graph. For the graph title parameter, enter “Currents”; for the vertical axis label, enter “kA”; and for the horizontal axis label, enter “Time (sec)”. Again, select three (3) for the “Maximum Number of Curve Labels in a Row”.

Select PROCEED to complete the properties modification.

3.4.2 Adding Text to a Page

Annotations may be added to a page using the "TEXT" item in the "CREATE" menu.



The initial position of the text is specified with input boxes. Once the text item has been created, it can be dragged visually to the desired location. The text size, orientation, and justification of the text string are specified using toggle boxes on the form.

Unlike graphs, text strings do not have a "Layout" feature to control the position of text items. Instead, a text string may be specified as being "On All Pages". If a text string that is "On All Pages" is moved or edited, all instances of the text string are affected.

Create the following text strings, using normal orientation and the "On All Pages" specification:

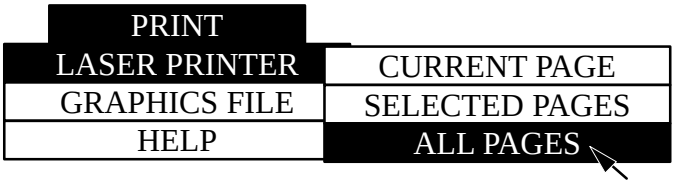
Text	From left	From bottom	Points	Justification
Voltage Divider	4.25	10.5	12	Centred
Page %# of %P	8	10.5	10	Right–justified
%c	0.5	10.5	10	Left–justified
%z	0.5	0.5	10	Left–justified

The "%_" characters in the above text strings are format codes, which are replaced by various pieces of information. "%#" is replaced by the current page number; "%P" by the total number of pages; "%c" by the current date & time; and "%z" is replaced by the company name as specified in the `$PSCAD_MASTER/database` file. For a complete list of format codes, select the ? box.

Which format codes are supported is a function of the operating system. For instance the HP425 does not support the "%T" short hand for "%H:%M:%S", but the latter sequence of codes is supported. Also, the result of some format codes is dependant on the "locale" specified for the machine. For instance, although the "%C" code specifies "Date & Time: locale's long format", it may produce only the date on some installations.

3.4.3 Printing the MultiPlot File

Once the document has been created, it can be printed by selecting “ALL PAGES” from the “LASER PRINTER” sub–menu under the “PRINT” menu.



3.4.4 Saving the MultiPlot File

The MultiPlot document, or batch file, may be saved by selecting “SAVE” under the “BATCH” menu. It is important to note that the information saved in the batch file include the page layout information, graph properties, and ***references*** to the curve data, not necessarily the curve data itself. A MultiPlot batch file is virtually useless without the output files referenced in the batch file, and the information file referenced in the batch file if channels were selected by name.

Curve data which is generated by MultiPlot, such as “calculator output” or “fourier analysis output” will be saved in the batch file, unless is it explicitly stored to an output file before the batch file is saved.

3.4.5 Updating Graphs

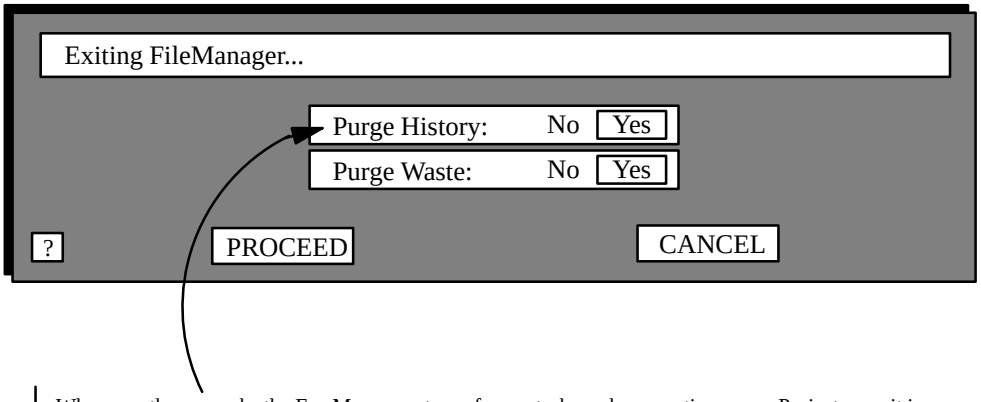
If at this point ICON was selected, MultiPlot would be reduced to an icon. The Draft module can then be re–opened, and the circuit modified, say by changing the 1 ohm resistors into 3000 μF capacitors. After compiling the circuit in Draft, RunTime can be re–opened and the simulation re–run, copying the output files back.

If MultiPlot were then re–opened, the message box would indicate that the curves are out–of–date. If the “UPDATE” option from the top of the module is selected, the curve information would be re–read, and the graphs would be updated.

Curve data which is generated by MultiPlot, such as “calculator output” or “fourier analysis output”, is not affected when “UPDATE” is used.

3.5 TERMINATING THE PSCAD SESSION

The PSCAD software can be terminated by selecting the QUIT option from the top menu area of the FILEMANAGER window. Any other PSCAD modules which were started from the FILEMANAGER will automatically be stopped.



Whenever the user asks the FILEMANAGER to perform a task, such as creating a new Project area, it is recorded in a History file. In order that this file not become too large it is automatically cleared unless the user should select “NO” for “Purge History”.

Any database entries which were deleted by the user are actually placed into a waste area so that they may be retrieved. If the user wishes to truly delete the files residing in the waste area then the user should select “YES” for “Purge Waste”. This option in greyed if no waste currently exists in the waste area.